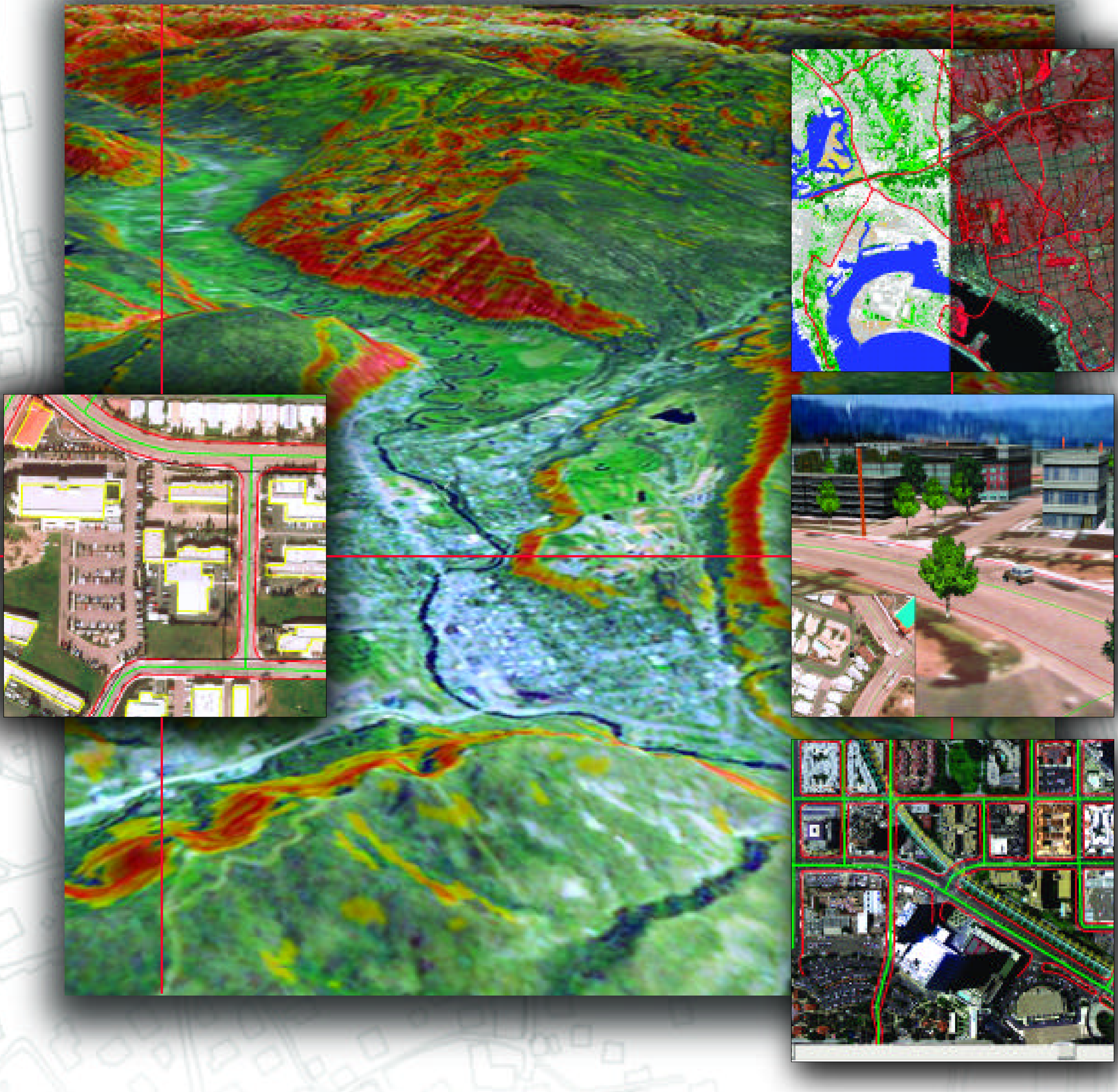
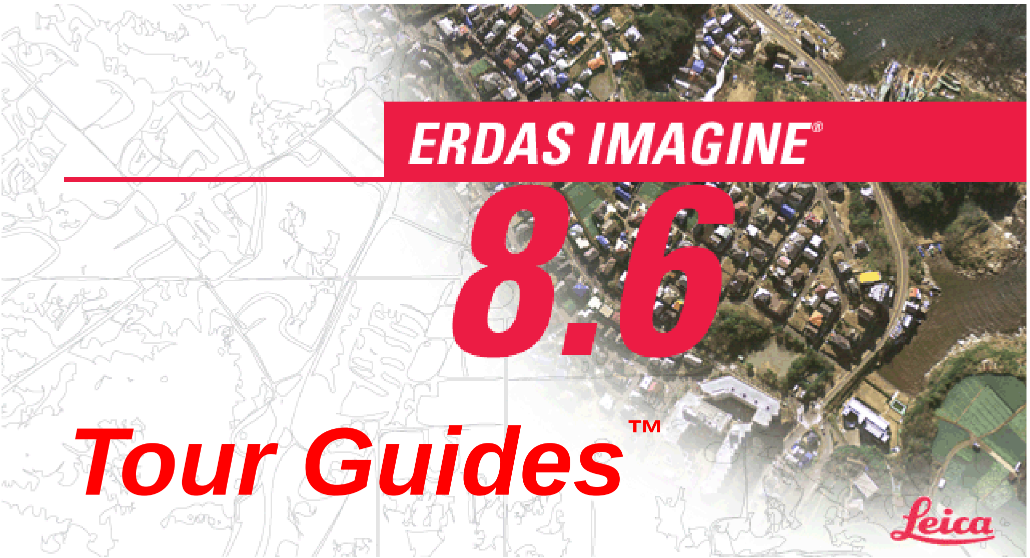
ERDAS IMAGINE Tour Guides™



ERDAS IMAGINE® 8.6



ERDAS IMAGINE®

8.6

Tour Guides™

Copyright © 1991-2002 by ERDAS® LLC. All Rights Reserved.

ERDAS Proprietary - Delivered under license agreement.
Copying and disclosure prohibited without express written permission from ERDAS LLC.

Leica Geosystems
GIS & Mapping Division
2801 Buford Highway, NE
Atlanta, Georgia 30329-2137 USA
www.gis.leica-geosystems.com
Toll Free: 877/463-7327
Phone: 404/248-9000
Fax: 404/248-9400
User Support: 404/248-9777

Acknowledgments

ERDAS would like to extend our thanks to the following data providers:

- U.S. Geological Survey, EROS Data Center, Sioux Falls, SD
- City of Oxford, OH - GeoOne, Inc.
- ESA/EURIMAGE, Rome, Italy - Distributed by NRSC

Warning

All information in this document, as well as the software to which it pertains, is proprietary material of ERDAS, Inc., and is subject to an ERDAS license and non-disclosure agreement. Neither the software nor the documentation may be reproduced in any manner without the prior written permission of ERDAS, Inc.

Specifications are subject to change without notice.

Trademarks

Copyright © 2002 Leica Geosystems, GIS & Mapping Division. All rights reserved. ERDAS, ERDAS IMAGINE, IMAGINE Essentials, IMAGINE Advantage, IMAGINE Professional, ViewFinder, IMAGIZER, IMAGINE OrthoBASE, and Stereo Analyst are registered trademarks; ERDAS MapSheets, ERDAS MapSheets Express, IMAGINE Radar Interpreter, IMAGINE OrthoRadar, IMAGINE StereoSAR DEM, IMAGINE IFSAR DEM, IMAGINE VirtualGIS, IMAGINE OrthoBASE Pro, IMAGINE Vector, IMAGINE Developers' Toolkit, IMAGINE Geospatial Light Table, IMAGINE GLT, ERDAS Field Guide, ERDAS IMAGINE Tour Guides, and CellArray are trademarks of ERDAS LLC. ERDAS is a wholly-owned subsidiary of Leica Geosystems and operates as the Geographic Imaging unit within Leica Geosystems' GIS & Mapping Division. Other brands and product names are the property of their respective owners. 10/02. Part No. EI86TG.

Table of Contents

Table of Contents iii

List of Tables xv

Preface xvii

- About This Manual xvii
- Example Data xvii
- Time Required xvii
- Documentation xvii
  - Digital Hardcopy Documentation xviii
  - On-Line Help Documentation xviii
  - Documentation Functions xx
- Conventions Used in This Book xx
- Getting Started xxi
  - ERDAS IMAGINE Icon Panel xxi
  - ERDAS IMAGINE Menu Bar xxi
  - Dialogs xxx
- More Information/Help xxx

Section I
IMAGINE Essentials™ 1

Chapter 1
Viewer 3

- Introduction 3
- Display Preferences 3
  - Check Band-to-Color Gun Assignments 3
  - Check Viewer Preferences 4
  - Check Preference Editor Help 4
  - View Category Help 5
- Display an Image 6
  - Display Options 8
- Utility Menu Options 10
  - Use Inquiry Functions 10
  - Change Inquire Cursor Style 11
  - Take Measurements 12

View Menu Options . . . 14
Arrange Layers . . . 14
Zoom . . . 16
Animated Zoom . . . 18
Box Zoom . . . 18
Real-time Zoom . . . 19
Display Two Images . . . 20
Link Viewers . . . 20
Compare Images . . . 21
Unlink Viewers . . . 21
Raster Menu Options . . . 22
Create an AOI Layer . . . 22
Adjust Image Contrast . . . 26
Use Piecewise Linear Stretches . . . 27
Manipulate Histogram . . . 28
Adjust Shift/Bias . . . 30
Use Mouse Linear Mapping . . . 31
Raster Editor . . . 33
Interpolate . . . 34
Fill with Constant Value . . . 35
Set Global Value . . . 36
Raster Attribute Editor . . . 36
Change Color Attribute . . . 36
Make Layers Transparent . . . 38
Manipulate CellArray Information . . . 39
Edit Column Properties . . . 40
Generate Statistics . . . 41
Select Criteria . . . 42
Generate Report . . . 43
Profile Tools . . . 46
Display Spectral Profile . . . 46
Display Spatial Profile . . . 49
View Surface Profile . . . 51
Image Drape . . . 52
Change Options . . . 54
Change Sun Position . . . 54
Dump Contents to Viewer . . . 55
Start Eye/Target . . . 55
Manipulate the Observer and Field of View . . . 56
Chapter 2
Image Catalog . . . 59
Introduction . . . 59
Set Catalog Preferences . . . 59
Create an Image Catalog . . . 61
Add Information . . . 61
Perform Graphical Queries . . . 62
Select Area . . . 62

Run Query 63
View Information 64
Modify Views 65

Archive Data 66

Chapter 3
Map Composer 69

Introduction 69

Create a Map 69

Start Map Composer 70

Prepare the Data Layers 72

Draw the Map Frame 72
Adjust the Size of the Map Frame 75
Adjust the Position of the Map Frame 76

Edit the Map Frame 77

Delete the Map Frame 77

Add a Neatline and Tick Marks 78
Change Text/Line Styles 80

Make Scale Bars 82

Create a Legend 84

Add a Map Title 86

Place a North Arrow 88

Write Descriptive Text 89
Save the Map Composition 90

Print the Map Composition 90

Edit Composition Paths 92

Chapter 4
Vector Querying and Editing 95

Introduction 95

Query Vector Data 95
Copy Vector Data 95
Display Vector Layers 97
Change Vector Properties 98
Display Attributes in the Viewer 100
View Attributes 100
Use the Marquee Tools to Select Features 102
Use the Criteria Function 103

Edit Vector Layers 104
Change Viewing Properties 105
Use Editing Tools and Commands 107
Create New Vector Layer 109
Export Zoning Attributes 111
Create Attributes 112
Create a Simple Shapefile Layer 114

Create a Shapefile Coverage . . . 115
Editing the Shapefile Layer . . . 117
Open a Personal Geodatabase . . . 120
Open An Enterprise Geodatabase . . . 123

Chapter 5
Classification . . . 127

Introduction . . . 127

Use Unsupervised Classification . . . 127
Generate Thematic Raster Layer . . . 128
Choose Processing Options . . . 129

Evaluate Classification . . . 130
Create Classification Overlay . . . 130
Open Raster Attribute Editor . . . 131
Analyze Individual Classes . . . 132

Chapter 6
Polynomial Rectification . . . 135

Introduction . . . 135

Rectify a Landsat Image . . . 135
Perform Image to Image Rectification . . . 135
Start GCP Tool . . . 137
Select GCPs . . . 140
Calculate Transformation Matrix from GCPs . . . 146
Digitize Check Points . . . 146
Resample the Image . . . 148
Verify the Rectification Process . . . 149

Rotate, Flip, or Stretch Images . . . 150

Chapter 7
Image Commands . . . 153

Introduction . . . 153

Image Commands . . . 153

Chapter 8
Import/Export . . . 159

Introduction . . . 159

Import a SPOT Scene . . . 159
Check Preview Options . . . 161
Check Import Options . . . 163

Export LAN Data . . . 165
Choose Export Options . . . 165

Create .tif Files . . . 167
Check the Classification . . . 169
Check Map Information . . . 170

Import Generic Binary Data . . . 171

Data View 172
Enter Image Dimensions 174
Save Options 175
Preview the Image 176

Get Image Information 176
Edit Image Information 176
View Image Information 177

Chapter 9
Batch Processing 179

Introduction 179

Set up/Start the Task Scheduler on NT and 2000 180

Use Batch with a UNIX System 182

Execute a Single File/Single Command 182

Execute Multiple Files/Single Command—Run Now 183
Set TIFF Image File Preferences 184
Start the Image Command Tool 184
Start the Batch Wizard 185

Execute Multiple Files/Single Command—Run Later 188
Add Multiple Files 190

Execute Multiple Files/Multiple Commands 193
Set Up 193
Run First Command 195
Run Next Command 196
Run Another Command 197
Create Variables 198
Modify Variables 198
Select Input Files 200
Set Start Time 201

Work with Variables 202
Create a New Variable 202
Prepare 202
Start Batch Wizard 203
Create a New Variable 204
Add Additional Files to the Batch Job 206
Start Processing 206
Check the Output Directory 207

Additional Information 207
Save/Load Options 207
Batch Job Files 208

Section II
IMAGINE Advantage™ 211

Chapter 10
Fourier Transform Editor 213

- **Introduction** . . . **213**
- **Create an .fft Layer** . . . **213**
  - Create FFT Output File . . . 215
- **Start the Fourier Transform Editor** . . . **216**
- **Edit Using Menu Options** . . . **219**
  - Use Low-Pass Filtering . . . 219
  - Select a Different Filter . . . 221
  - Save the File . . . 221
  - Apply an Inverse Fourier Transformation . . . 222
  - Apply Other Filters . . . 223
  - Use a Circular Mask . . . 226
  - Use a Rectangular Mask . . . 228
  - Use a Wedge Mask . . . 229
- **Edit Using Mouse-Driven Tools** . . . **231**
  - Use Low-Pass Filtering . . . 231
  - Use High-Pass Filtering . . . 231
  - Apply a Wedge Mask . . . 233
  - Combine Edits . . . 234

Chapter 11
Image Interpreter . . . **237**

- **Introduction** . . . **237**
- **Subsetting an Image** . . . **237**
  - Selecting an AOI to Subset . . . 238
  - Subsetting an Image Without Snapping . . . 239
  - Displaying the Subset with the Original Data . . . 242
  - Subsetting an Image With Snap to Raster . . . 244
  - Viewing the Snapped Subset . . . 245
- **Apply Spatial Enhancement** . . . **246**
  - Apply Convolution . . . 246
  - Select Input/Output Files . . . 247
  - Select Kernel . . . 247
  - Check the File . . . 248
  - Apply Crisp . . . 249
  - Use the View Option . . . 250
  - View Results . . . 251
- **Apply Radiometric Enhancement** . . . **252**
  - Apply Brightness Inversion . . . 253
  - Reverse . . . 254
  - View Changes . . . 254
- **Apply Spectral Enhancement** . . . **256**
  - Use Tasseled Cap . . . 256
  - Check Results . . . 258
  - Use the Indices Function . . . 259
  - Choose RGB to IHS . . . 260
  - Choose IHS to RGB . . . 261
  - View the Results . . . 261

Chapter 12

Orthorectification 265

- **Introduction 265**
- **Rectify a Camera Image 265**
  - Perform Image to Image Rectification 265
  - Check for Map Model 266
  - Redisplay the file 267
  - Perform Geometric Correction 268
  - Set Camera Model Properties 269
  - Edit Fiducials 269
  - Enter Film Coordinates 272
  - Change Projection 273
  - Name the Geometric Model 274
  - Start the GCP Tool and compute RMS Error 275
  - Choose Your Path 276
  - Resample the Image 277
  - Calibrate the Image 278

Chapter 13

Terrain Surface Interpolation 281

- **Introduction 281**
- **Create a Surface 281**
  - Import an ASCII File 282
  - Perform Surfacing 285
  - Display the Surface 286

Chapter 14

Mosaic 289

- **Introduction 289**
- **Mosaic Using Air Photo Images 289**
  - Set Input Images 292
  - Create a Template 294
  - Identify Areas of Intersection 296
  - Draw a Cutline 298
  - Define Output Images 301
  - Run the Mosaic 302
  - Display the Mosaic 303
- **Mosaic Using LANDSAT Images 304**
  - Display Input Images 304
  - Add Images for Mosaic 306
  - Stack Images 308
  - Match Images 308
  - Run the Mosaic 309
  - Display Output Image 310
- **Mosaic Color Balancing Using Color Infrared Aerial Photos 312**
  - Add Images for Mosaic 314
  - Set Exclude Areas 315
  - Mosaic Color Balancing 317

Match Images 320
Run The Mosaic 321
Display Output Image 321

Chapter 15
Viewshed Analysis 323

Introduction 323

Create a Viewshed 323
Add the Raster Image 324
Start an Image Drape Viewer and Set the Level of Detail 324
Start the Viewshed Analysis Tool 325
Add First Observer 327
Add Another Observer 328
Link the Viewers and Set Eye and Target Positions 330
Save the Viewshed 331
Query Viewshed Data 332
Add Area Column to the CellArray 333
Query the Viewshed Layer 334
Finish 335

Section III
IMAGINE Professional™ 337

Chapter 16
Spatial Modeler 339

Introduction 339

Start Model Maker 340

Create Sensitivity Layer 340
Define Input Slope Layer 342
Display Slope Layer 343
Select Area to Use 344
Recode Classes 345
Define Input Flood Plain Layer 347
Define Input Land Cover Layer 347
Define Function 349
Define Output Raster Layer 350
Save and Run the Model 351
Run the Model 352

Enhance SPOT Data 352
Define Input SPOT Layer 354
Define Input Convolution Kernel 354
Define Function 355
Define Output Raster Layer 356
Save and Run the Model 357
Run the Model 357
Combine Models 357

Combine Sensitivity Layer with SPOT Data 358

Define Input Scalar 359
Define Function 359
Define Output Raster Layer 361
Save and Run the Model 363

Display New Layer 363
Adjust Colors 364

Test the Output 367

Add Annotation to a Model 368
Add a Title 368
Format Text 368
Add Text to a Function Graphic 369
Format Text 369
Add Text to Other Graphics 369

Generate a Text Script 371

Print the Model 373

Apply the Criteria Function 375
Evaluate Training Samples 375
Define Input Raster Layers 377
Define Criteria 377
Define Output Raster Layer 379
Save the Model 380

Minimizing Temporary Disk Usage 381
Set Preferences 381

Making Your Models Usable by Others 382
Prompt User 382
Providing a User Interface to Your Model 382
Open an Existing Model 383
Edit the Model 385
Edit the EML 387
Set Session Commands 391
Check the Results 392
Use the Swipe Utility 393
Check the spots.img image 394

Using Vector Layers in Your Model 395
Vector Layers as a Mask 395
Add Attributes to Vector Layers 408

Debug Your Model 415
Eliminate Incomplete Definition 415
Eliminate Object type Mismatch 428
Eliminate Division by Zero 443
Use AOIs in Processing 448
Using the Swipe Utility 453

Chapter 17
Advanced Classification 455

Introduction 455
Supervised vs. Unsupervised Classification 455

Perform Supervised Classification 456
Define Signatures using Signature Editor 456
Use Tools to Evaluate Signatures 468
Perform Supervised Classification 479

Perform Unsupervised Classification 483
Generate Thematic Raster Layer 484

Evaluate Classification 486
Create Classification Overlay 486
Analyze Individual Classes 489
Use Thresholding 491
Use Accuracy Assessment 496

Using the Grouping Tool 500
Setting Up a Class Grouping Project 500
Collecting Class Groups 504
Using the Ancillary Data Tool 512
Coloring the Thematic Table 518

Chapter 18
Frame Sampling Tools 521

Introduction 521
Remote Sensing and Frame Sampling 521
Frame Sampling Tools 521

Setting Up the Sampling Project 522
Create a New Sampling Project 522

Root Level Functions 524

Tile Level Functions 528
Selecting the Samples 536

Sample Level Functions 540
Dot Grid Interpretation 542
Final Analysis Wizard 552

Chapter 19
IMAGINE Expert Classifier™ 559

Introduction 559

Create a Knowledge Base 559
Set Up the Output Classes 559
Enter Rules for the Hypothesis 562
Add an Intermediate Hypothesis 566
Copy and Edit 568
Test the Knowledge Base 570
Data 575
Methodology 575
Open a Knowledge Base 576
Examine the Knowledge Base 577
Derive Slope Values 578
Build Hypotheses 581
Set ANDing Criteria 583
Check Other Hypotheses 584

    - Introduce Spatial Logic to the Knowledge Base . . . 585
    - Check Buildings Hypothesis . . . 587
    - Identify Choke Points . . . 588
    - Run the Expert Classification . . . 591
    - Evaluate River Areas . . . 593
    - Use Pathway Feedback . . . 594

Section IV
IMAGINE Radar Interpreter™ . . . 597

Chapter 20
IMAGINE Radar Interpreter™ . . . 599

- **Introduction . . . 599**
- **Suppress Speckle Noise . . . 599**
    - Calculate Coefficient of Variation . . . 603
    - Run Speckle Suppression Function . . . 604
    - Use Histograms to Evaluate Images . . . 606
- **Enhance Edges . . . 608**
- **Enhance Image . . . 611**
    - Wallis Adaptive Filter . . . 612
    - Apply Sensor Merge . . . 614
    - Apply Texture Analysis . . . 618
- **Adjust Brightness . . . 621**
- **Adjust Slant Range . . . 622**

Section V
IMAGINE Vector™ . . . 625

Chapter 21
IMAGINE Vector™ . . . 627

- **Introduction . . . 627**
- **Copy Vector Data . . . 628**
- **Manipulate Info Files . . . 630**
- **Change Vector Symbology . . . 637**
    - Add Pattern Polygon Fill . . . 640
- **Build Topology . . . 643**
- **Clean Vector Layer . . . 645**

Index . . . 647

List of Tables

Table P-1: ERDAS IMAGINE Documentation Functions xx
Table P-2: Session Menu Options xxii
Table P-3: Main Menu Options xxiii
Table P-4: Tools Menu Options xxiv
Table P-5: Utility Menu Options xxvi
Table P-6: Help Menu Options xxvii
Table 2-1: Image Catalog Preferences 59
Table 12-1: Film X and Film Y Coordinates 272
Table 16-1: Class Values for n3_landcover_RC 348
Table 16-2: Conditional Statement Class Values 350
Table 16-3: Training Samples of Chaparral and Riparian Land Cover 376
Table 16-4: Complete Criteria Table 379
Table 20-1: Coefficient of Variation Values for Look-averaged Radar Scenes 603
Table 20-2: Speckle Suppression Parameters 605
Table 20-3: Filtering Sequence 606

List of Tables

Preface

About This Manual

The *ERDAS IMAGINE Tour Guides*™ manual is a compilation of tutorials designed to help you learn how to use ERDAS IMAGINE® software. This is a comprehensive manual, representing ERDAS IMAGINE and its add-on modules. Each guide takes you step-by-step through an entire process. The tour guides are not intended to tell you everything there is to know about any one topic, but to show you how to use some of the basic tools you need to get started.

This manual serves as a handy reference that you can refer to while using ERDAS IMAGINE for your own projects. Included is a comprehensive index, so that you can reference particular information later.

There are several sections to this manual. These sections are based upon the way in which ERDAS IMAGINE is packaged. The following sections are composed of the tour guides you go through in a step by step fashion to learn detailed information about the various ERDAS IMAGINE functions.

- Section I—IMAGINE Essentials™
- Section II—IMAGINE Advantage™
- Section III—IMAGINE Professional™
- Section IV—IMAGINE Radar Interpreter™
- Section V—IMAGINE Vector™

Example Data

Data sets are provided with the software so that your results match those in the tour guides. The data used in the tour guides are in the <IMAGINE_HOME>/examples directory. <IMAGINE_HOME> is the variable name of the directory where ERDAS IMAGINE resides. When accessing data files, you must replace <IMAGINE_HOME> with the name of the directory where ERDAS IMAGINE is loaded on your system.

Time Required

Each individual tour guide takes a different amount of time to complete, depending upon the options you choose and the length of the tour guide. The approximate completion time is stated in the introduction to each tour guide.

Documentation

This manual is part of a suite of on-line documentation that you receive with ERDAS IMAGINE software. There are two basic types of documents, digital hardcopy documents which are delivered as PDF files suitable for printing or on-line viewing, and On-Line Help Documentation, delivered as HTML files.

Digital Hardcopy Documentation

The ERDAS IMAGINE Digital Hardcopy Documentation is designed to provide comprehensive information about a particular concept or to walk you through complicated steps in a process like the Installation of IMAGINE or Advanced Classification. The Digital Hardcopy Documentation also contains programming reference material, such as the ERDAS Macro Language Reference Manual which helps you design your own IMAGINE dialogs. These documents may be found in the <IMAGINE_HOME>/help/hardcopy folder.

To read the IMAGINE Digital Hardcopy Documentation, you must install Adobe Acrobat Reader 4.0 or higher and follow all installation instructions provided by the software, especially those regarding internet browser and Acrobat integration.

Following is a list of Digital Hardcopy Documentation that is available with ERDAS IMAGINE software:

- *ERDAS Field Guide™* (FieldGuide.pdf)
- *ERDAS IMAGINE Configuration Guide* (ConfigGuide.pdf)
- *ERDAS IMAGINE Tour Guides™* (TourGuide.pdf)
- ERDAS Macro Language Reference Manual (EML.pdf)
- ERDAS Spatial Modeler Language Reference Manual (SML.pdf)
- ERDAS IMAGINE Read Me First document
- ERDAS IMAGINE V8.6 Release Notes
- FLEXlm End Users Guide (enduser.pdf)
- What’s New in ERDAS IMAGINE V8.6

On-Line Help Documentation

The IMAGINE On-Line Help Documentation is set up as a network of HTML files that are displayed in your default internet browser and provide quick, informative chunks of information on all of the IMAGINE dialogs, as well as additional explanatory notes and diagrams. This HTML database includes JavaScript applets that provide an expanding and collapsing Table of Contents, Index, and Full Text Search utilities. To use these applets you must have installed a compliant browser (Netscape 4.7 or Internet Explorer 5.5 or higher are greatly recommended) and that you enable Java scripting in your browser properties.

Following is a list of on-line manuals that can be found in the On-Line Help in ERDAS IMAGINE software. This list may change depending on your software package and add-on modules you have purchased:

- Introduction to ERDAS IMAGINE On-Line Help
- Annotation On-Line Manual
- AOI On-Line Manual
- Digital Point Positioning Database Workstation
- Classification On-Line Manual
- Expert Classifier On-Line Manual
- Stereo Analyst On-Line Manual
- HyperSpectral On-Line Manual
- IMAGINE Command and Function Syntax On-Line Manual

- IMAGINE Interface On-Line Manual
- Image Catalog On-Line Manual
- Image Interpreter On-Line Manual
- Import/Export On-Line Manual
- Importing Native Formats On-Line Manual
- Map Composer On-Line Manual
- NITF
- OrthoBASE On-Line Manual
- OrthoBASE Pro On-Line Manual
- Radar Mapping Suite On-Line Manual
- IFSAR DEM on-Line Manual
- StereoSAR DEM On-Line Manual
- OrthoRadar On-Line Manual
- Generic SAR Node On-Line Manual
- Rectification On-Line Manual
- Session On-Line Manual
- Preferences On-Line Manual
- Spatial Modeler On-Line Manual
- Spectral Analysis
- Tools and Utilities On-Line Manual
- Vector On-Line Manual
- Viewer On-Line Manual
- Viewer Raster Tools On-Line Manual
- Virtuality On-Line Manual
- Graphical Models Reference Guide
- DLL Reference Guide
- Imagizer On-Line Manual
- Creating On-Line Help for IMAGINE
- Appendices

Documentation Functions

The following table depicts the different types of information you can extract from ERDAS IMAGINE documentation.

Table P-1: ERDAS IMAGINE Documentation Functions

If you want to...	**Read...**
Install ERDAS IMAGINE	*ERDAS IMAGINE Release Notes*, then *ERDAS IMAGINE Configuration Guide*
Set up hardware for use with ERDAS IMAGINE	*ERDAS IMAGINE Configuration Guide*
Learn to use ERDAS IMAGINE	*ERDAS IMAGINE Tour Guides*
Learn about GIS and image processing theory	*ERDAS Field Guide*
See what a particular dialog does	*On-Line Help*
Get quick information for a button or function	*On-Line Help* or *Status Bar Help*
Learn how to most effectively use the On-Line Help system	*Introduction to On-Line Help in the On-Line Help*
Learn more about the Image Interpreter functions	*ERDAS IMAGINE Tour Guides*
Use the Spatial Modeler Language to write models	*ERDAS Spatial Modeler Language Reference Manual*
Customize the ERDAS IMAGINE graphical user interface (GUI)	*ERDAS Macro Language Reference Manual*
Write custom application programs within ERDAS IMAGINE	*ERDAS Developers' Toolkit On-Line Manual*

Conventions Used in This Book

In ERDAS IMAGINE, the names of menus, menu options, buttons, and other components of the interface are shown in bold type. For example:

"In the Select Layer To Add dialog, select the **Fit to Frame** option."

When asked to use the mouse, you are directed to click, Shift-click, middle-click, right-click, hold, drag, etc.

- click—designates clicking with the left mouse button.
- Shift-click—designates holding the Shift key down on your keyboard and simultaneously clicking with the left mouse button.
- middle-click—designates clicking with the middle mouse button.
- right-click—designates clicking with the right mouse button.
- hold—designates holding down the left (or right, as noted) mouse button.
- drag—designates dragging the mouse while holding down the left mouse button.

The following paragraphs are used throughout the ERDAS IMAGINE documentation:

These paragraphs contain strong warnings or important tips.

These paragraphs direct you to the ERDAS IMAGINE software function that accomplishes the described task.

These paragraphs lead you to other areas of this book or other ERDAS® manuals for additional information.

NOTE: Notes give additional instruction.

Shaded Boxes

Shaded boxes contain supplemental information that is not required to execute the steps of a tour guide, but is noteworthy. Generally, this is technical information.

Getting Started

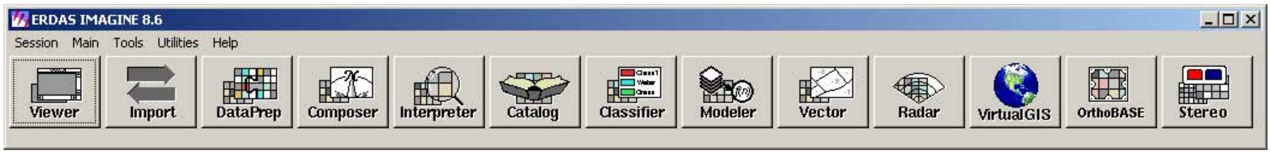
To start ERDAS IMAGINE, type the following in a UNIX command window: **imagine**, or select **ERDAS IMAGINE 8.6** from the **Start** menu.

ERDAS IMAGINE begins running; the icon panel automatically opens.

ERDAS IMAGINE Icon Panel

The ERDAS IMAGINE icon panel contains icons and menus for accessing ERDAS IMAGINE functions. You have the option (through the **Session | Preferences** menu) to display the icon panel horizontally across the top of the screen or vertically down the left side of the screen. The default is a horizontal display.

The icon panel that displays on your screen looks similar to the following:



The various icons that are present on your icon panel depend on the components and add-on modules you have purchased with your system.

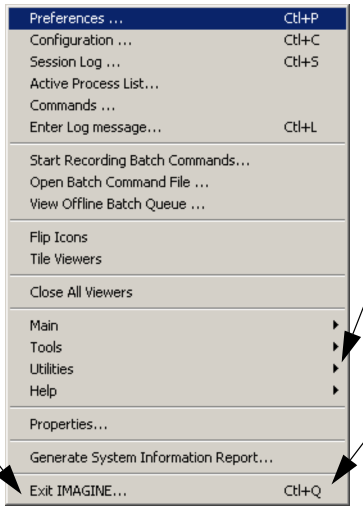
ERDAS IMAGINE Menu Bar

The menus on the ERDAS IMAGINE menu bar are: **Session**, **Main**, **Tools**, **Utilities**, and **Help**. These menus are described in this section.

NOTE: Any items which are unavailable in these menus are shaded and inactive.

Session Menu

1. Click on the word **Session** in the upper left corner of the ERDAS IMAGINE menu bar. The **Session** menu opens:


Preferences ... Ctl+P
Configuration ... Ctl+C
Session Log ... Ctl+S
Active Process List...
Commands ...
Enter Log message... Ctl+L
Start Recording Batch Commands...
Open Batch Command File ...
View Offline Batch Queue ...
Flip Icons
Tile Viewers
Close All Viewers
Main
Tools
Utilities
Help
Properties...
Generate System Information Report...
Exit IMAGINE... Ctl+Q

Click here to end the ERDAS IMAGINE session

These menus are identical to the ones on the icon panel.

You can also place your cursor anywhere in the icon panel and press Ctrl-Q to exit ERDAS IMAGINE

The following table contains the **Session** menu selections and their functionalities:

Table P-2: Session Menu Options

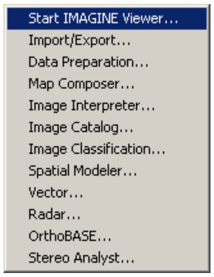
Selection	Functionality
Preferences	Set individual or global default options for many ERDAS IMAGINE functions (Viewer, Map Composer, Spatial Modeler, etc.).
Configuration	Configure peripheral devices for ERDAS IMAGINE.
Session Log	View a real-time record of ERDAS IMAGINE messages and commands, and to issue commands.
Active Process List	View and cancel currently active processes running in ERDAS IMAGINE.
Commands	Open a command shell, in which you can enter commands to activate or cancel processes.
Enter Log Message	Insert text into the Session Log.
Start Recording Batch Commands	Open the Batch Wizard. Collect commands as they are generated by clicking the **Batch** button that is available on many ERDAS IMAGINE dialogs.
Open Batch Command File	Open a Batch Command File (*.bcf) you have saved previously.
View Offline Batch Queue	Open the Scheduled Batch Job list dialog, which gives information about pending batch jobs.
Flip Icons	Specify horizontal or vertical icon panel display.

Table P-2: Session Menu Options (Continued)

Selection	Functionality
Tile Viewers	Rearrange two or more Viewers on the screen so that they do not overlap.
Close All Viewers	Close all Viewers that are currently open.
Main	Access a menu of tools that corresponds to the icons along the ERDAS IMAGINE icon bar.
Tools	Access a menu of tools that allow you to view and edit various text and image files.
Utilities	Access a menu of utility items that allow you to perform general tasks in ERDAS IMAGINE.
Help	Access the ERDAS IMAGINE On-Line Help.
Properties	Display the ERDAS IMAGINE Properties dialog where system, environment and licensing information is available.
Generate System Information Report	Provides a mechanism for printing essential IMAGINE operating system parameters.
Exit IMAGINE	Exit the ERDAS IMAGINE session (keyboard shortcut: Ctrl-Q).

Main Menu

2. Click on the word **Main** in the ERDAS IMAGINE menu bar. The **Main** menu opens


Start IMAGINE Viewer...
Import/Export...
Data Preparation...
Map Composer...
Image Interpreter...
Image Catalog...
Image Classification...
Spatial Modeler...
Vector...
Radar...
OrthoBASE...
Stereo Analyst...

The following table contains the **Main** menu selections and their functionalities:

.

Table P-3: Main Menu Options

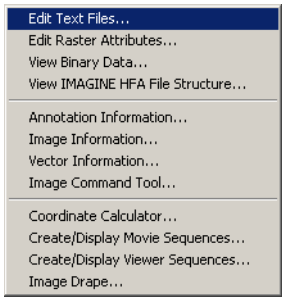
Selection	Functionality
Start IMAGINE Viewer	Start an empty Viewer.
Import/Export	Open the Import/Export dialog.
Data Preparation	Open the **Data Preparation** menu.
Map Composer	Open the **Map Composer** menu.

Table P-3: Main Menu Options (Continued)

Selection	Functionality
Image Interpreter	Open the **Image Interpreter** menu.
Image Catalog	Open the Image Catalog dialog.
Image Classification	Open the **Classification** menu.
Spatial Modeler	Open the **Spatial Modeler** menu.
Vector	Open the **Vector Utilities** menu.
Radar	Open the **Radar** menu.
VirtualGIS	Open the **VirtualGIS** menu.
OrthoBASE	Open the OrthoBASE Startup dialog.
Stereo Analyst	Open the Stereo Analyst Workspace.

Tools Menu

3. Click on the word **Tools** in the ERDAS IMAGINE menu bar. The **Tools** menu opens:


Edit Text Files...
Edit Raster Attributes...
View Binary Data...
View IMAGINE HFA File Structure...
Annotation Information...
Image Information...
Vector Information...
Image Command Tool...
Coordinate Calculator...
Create/Display Movie Sequences...
Create/Display Viewer Sequences...
Image Drape...

The following table contains the **Tools** menu selections and their functionalities:

Table P-4: Tools Menu Options

Selection	Functionality
Edit Text Files	Create and edit ASCII text files.
Edit Raster Attributes	Edit raster attribute data.
View Binary Data	View the contents of binary files in a number of different ways.

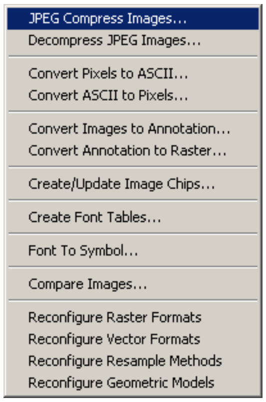
Table P-4: Tools Menu Options (Continued)

Selection	Functionality
View IMAGINE HFA File Structure	View the contents of the ERDAS IMAGINE hierarchical files.
Annotation Information	View information for annotation files, including number of elements and projection information.
Image Information	Obtain full image information for a selected ERDAS IMAGINE raster image.
Vector Information	Obtain full image information for a selected ERDAS IMAGINE vector coverage.
Image Command Tool	Open the Image Command dialog.
Coordinate Calculator	Transform coordinates from one spheroid or datum to another.
Create/Display Movie Sequences	View a series of images in rapid succession.
Create/Display Viewer Sequences	View a series of images saved from the Viewer.
Image Drape	Create a perspective view by draping imagery over a terrain DEM.
DPPDB Workstation	Start the Digital Point Positioning DataBase Workstation (if installed).
View EML ScriptFiles[a]	Open the EML View dialog, which enables you to view, edit, and print ERDAS IMAGINE dialogs.

[a] UNIX only.

Utilities Menu

4. Click on **Utilities** on the ERDAS IMAGINE menu bar. The **Utilities** menu opens:


JPEG Compress Images...
Decompress JPEG Images...
Convert Pixels to ASCII...
Convert ASCII to Pixels...
Convert Images to Annotation...
Convert Annotation to Raster...
Create/Update Image Chips...
Create Font Tables...
Font To Symbol...
Compare Images...
Reconfigure Raster Formats
Reconfigure Vector Formats
Reconfigure Resample Methods
Reconfigure Geometric Models

The following table contains the **Utilities** menu selections and their functionalities:

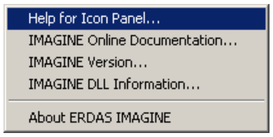
Table P-5: Utility Menu Options

Selection	Functionality
JPEG Compress Images	Compress raster images using the JPEG compression technique and save them in an ERDAS IMAGINE format.
Decompress JPEG Images	Decompress images compressed using the JPEG Compress Images utility.
Convert Pixels to ASCII	Output raster data file values to an ASCII file.
Convert ASCII to Pixels	Create an image from an ASCII file.
Convert Images to Annotation	Convert a raster image to polygons saved as ERDAS IMAGINE annotation (.ovr).
Convert Annotation to Raster	Convert an annotation file containing vector graphics to a raster image file.
Create/Update Image Chips	Provide a direct means of creating chips for one or more images.
Mount/Unmount CD-ROM[a]	Mount and unmount a CD-ROM drive.
Create Lowercase Parallel Links[a]	Make a set of links to items on CD for systems that convert CD paths to uppercase.
Create Font Tables	Create a map of characters in a particular font.
Font to Symbol	Create a symbol library to use as annotation characters from an existing font.
Compare Images	Open Image Compare dialog. Compare layers, raster, map info, etc.
Reconfigure Raster Formats	Start a DLL to reconfigure raster formats.
Reconfigure Vector Formats	Start a DLL to reconfigure vector formats.
Reconfigure Resample Methods	Start a DLL to reconfigure resampling methods.
Reconfigure Geometric Models	Start a DLL to reconfigure the geometris models.

[a] UNIX only.

Help Menu

5. Select **Help** from the ERDAS IMAGINE menu bar. The **Help** menu opens.


Help for Icon Panel...
IMAGINE Online Documentation...
IMAGINE Version...
IMAGINE DLL Information...
About ERDAS IMAGINE

NOTE: The **Help** *menu is also available from the* **Session** *menu.*

The following table contains the **Help** menu selections and their functionalities:

Table P-6: Help Menu Options

Selection	Functionality
Help for Icon Panel	View the On-Line Help for the ERDAS IMAGINE icon panel.
IMAGINE Online Documentation	Access the root of the On-Line Help tree.
IMAGINE Version	View which version of ERDAS IMAGINE you are running.
IMAGINE DLL Information	Display and edit DLL class information and DLL instance information.
About ERDAS IMAGINE	Open ERDAS IMAGINE Credits.

6. From the **Help** menu, select **Help for Icon Panel**. The following Help page displays in your default internet browser:

Back button
Click to return to the last topic viewed

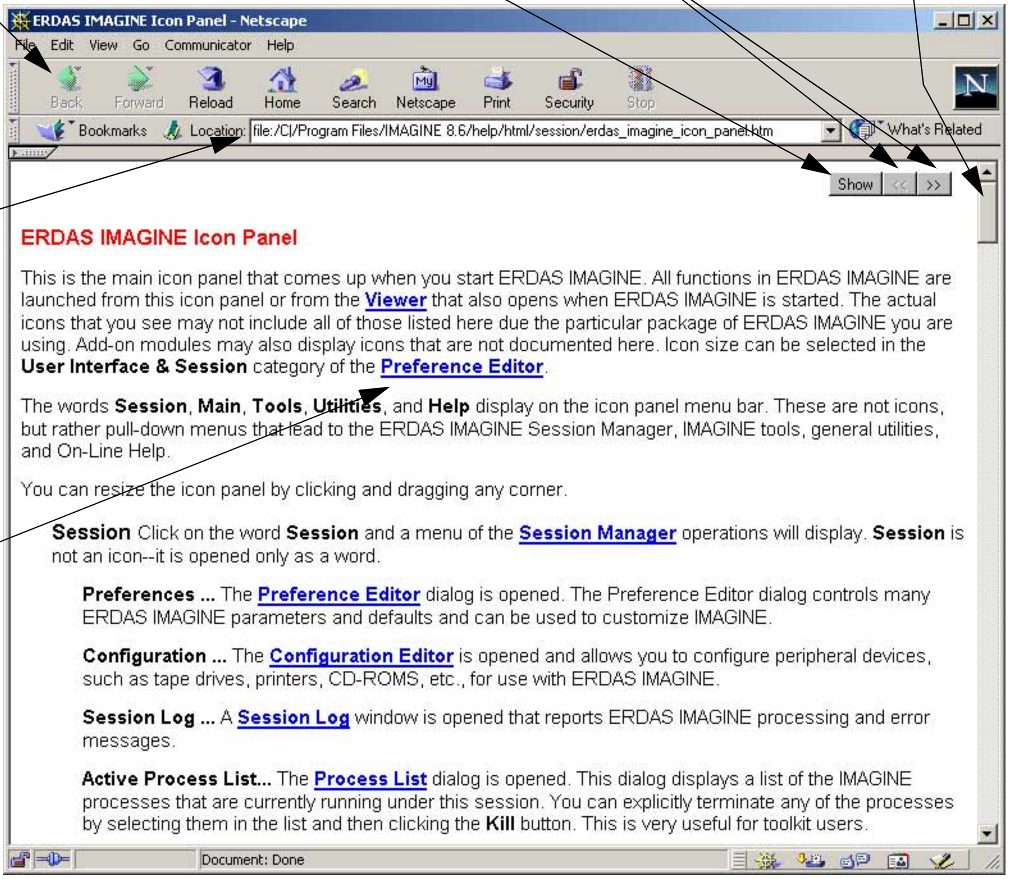
Show/Hide button
Displays or hides the navigation pane

Browse buttons
Click to move to the next or previous topic in the browse sequence

Slider bar
Use the slider bar to scroll through long help topics

Location bar
Displays the location of the current help file

Hyperlinks
Click on the blue* underlined words to link to other help topics



ERDAS IMAGINE Icon Panel

This is the main icon panel that comes up when you start ERDAS IMAGINE. All functions in ERDAS IMAGINE are launched from this icon panel or from the Viewer that also opens when ERDAS IMAGINE is started. The actual icons that you see may not include all of those listed here due the particular package of ERDAS IMAGINE you are using. Add-on modules may also display icons that are not documented here. Icon size can be selected in the **User Interface & Session** category of the Preference Editor.

The words **Session**, **Main**, **Tools**, **Utilities**, and **Help** display on the icon panel menu bar. These are not icons, but rather pull-down menus that lead to the ERDAS IMAGINE Session Manager, IMAGINE tools, general utilities, and On-Line Help.

You can resize the icon panel by clicking and dragging any corner.

Session Click on the word **Session** and a menu of the Session Manager operations will display. **Session** is not an icon--it is opened only as a word.

Preferences ... The Preference Editor dialog is opened. The Preference Editor dialog controls many ERDAS IMAGINE parameters and defaults and can be used to customize IMAGINE.

Configuration ... The Configuration Editor is opened and allows you to configure peripheral devices, such as tape drives, printers, CD-ROMS, etc., for use with ERDAS IMAGINE.

Session Log ... A Session Log window is opened that reports ERDAS IMAGINE processing and error messages.

Active Process List... The Process List dialog is opened. This dialog displays a list of the IMAGINE processes that are currently running under this session. You can explicitly terminate any of the processes by selecting them in the list and then clicking the **Kill** button. This is very useful for toolkit users.

***** blue is the default link color; your system may be different

The ERDAS IMAGINE On-Line Help (OLH) system is a collection of on-line manuals that functions just like an internet website. The OLH system includes help for all of the dialogs in ERDAS IMAGINE, as well as in-depth Help files that provide more details about a process. Each manual covers a specific topic. These manuals can be printed for convenience. The individual html files may also be bookmarked for quick reference.

7. Click the Show button at the upper right corner of the help topic. The Navigation Pane displays in the left portion of your browser.

Contents
Click on this tab to view the Table of Contents

Index
Click on this tab to look up help topics by keyword

Search
Click on this tab to perform a Full Text Search for a word or phrase

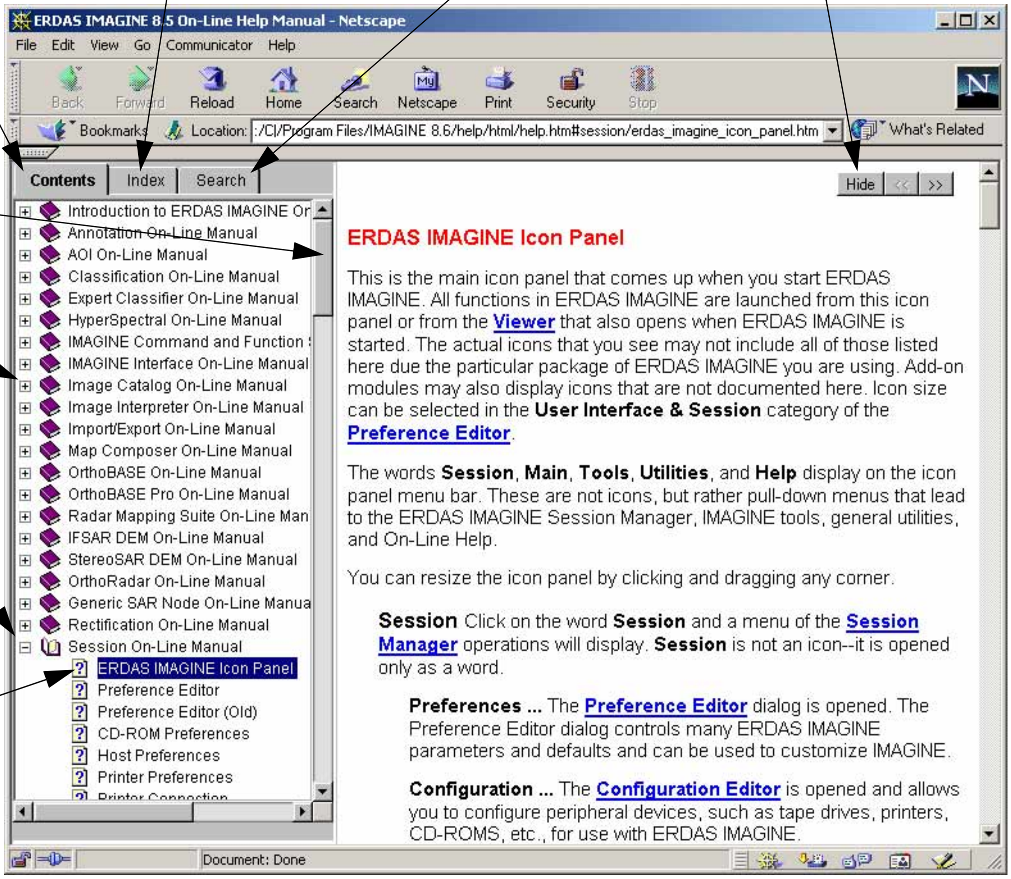
Show/Hide button
Displays or hides the navigation pane

Use the slider bars to scroll through the navigation pane

Click the + to open a book and browse its topics

Click the - to close a book and hide its topics

Click once to select a topic. Double-click to open a topic. The currently displayed topic is highlighted in the Table of Contents.


ERDAS IMAGINE 8.5 On-Line Help Manual - Netscape
File
Edit
View
Go
Communicator
Help
Back
Forward
Reload
Home
Search
Netscape
Print
Security
Stop
Bookmarks
Location: :/C|/Program Files/IMAGINE 8.6/help/html/help.htm#session/erdas_imagine_icon_panel.htm
What's Related
Contents
Index
Search
Hide
<<
>>
Introduction to ERDAS IMAGINE On
Annotation On-Line Manual
AOI On-Line Manual
Classification On-Line Manual
Expert Classifier On-Line Manual
HyperSpectral On-Line Manual
IMAGINE Command and Function
IMAGINE Interface On-Line Manual
Image Catalog On-Line Manual
Image Interpreter On-Line Manual
Import/Export On-Line Manual
Map Composer On-Line Manual
OrthoBASE On-Line Manual
OrthoBASE Pro On-Line Manual
Radar Mapping Suite On-Line Man
IFSAR DEM On-Line Manual
StereoSAR DEM On-Line Manual
OrthoRadar On-Line Manual
Generic SAR Node On-Line Manua
Rectification On-Line Manual
Session On-Line Manual
ERDAS IMAGINE Icon Panel
Preference Editor
Preference Editor (Old)
CD-ROM Preferences
Host Preferences
Printer Preferences
Printer Connection
Document: Done

ERDAS IMAGINE Icon Panel

This is the main icon panel that comes up when you start ERDAS IMAGINE. All functions in ERDAS IMAGINE are launched from this icon panel or from the Viewer that also opens when ERDAS IMAGINE is started. The actual icons that you see may not include all of those listed here due the particular package of ERDAS IMAGINE you are using. Add-on modules may also display icons that are not documented here. Icon size can be selected in the **User Interface & Session** category of the Preference Editor.

The words **Session**, **Main**, **Tools**, **Utilities**, and **Help** display on the icon panel menu bar. These are not icons, but rather pull-down menus that lead to the ERDAS IMAGINE Session Manager, IMAGINE tools, general utilities, and On-Line Help.

You can resize the icon panel by clicking and dragging any corner.

Session Click on the word **Session** and a menu of the Session Manager operations will display. **Session** is not an icon--it is opened only as a word.

Preferences ... The Preference Editor dialog is opened. The Preference Editor dialog controls many ERDAS IMAGINE parameters and defaults and can be used to customize IMAGINE.

Configuration ... The Configuration Editor is opened and allows you to configure peripheral devices, such as tape drives, printers, CD-ROMS, etc., for use with ERDAS IMAGINE.

8. Click on the + beside the **Viewer On-Line Manual** in the **Contents** tab to view all of the topics in the **Viewer** manual.

9. Double-click on the **Viewer** topic.

10. The first page of the Viewer Manual is displayed. Use the << and >> browse buttons to jump from topic to topic and the scroll bars to page through a topic. Click on any hyperlinked text to jump to that topic. Click the **Back** button to return to the point from which you jumped.

11. Use the **Contents**, **Index**, and **Search** tabs when looking for a specific title, subject, or word or phrase.

12. Select **File | Exit** from the On-Line Help file menu bar when you are finished reading On-Line Help. The Help window is closed.

Dialogs

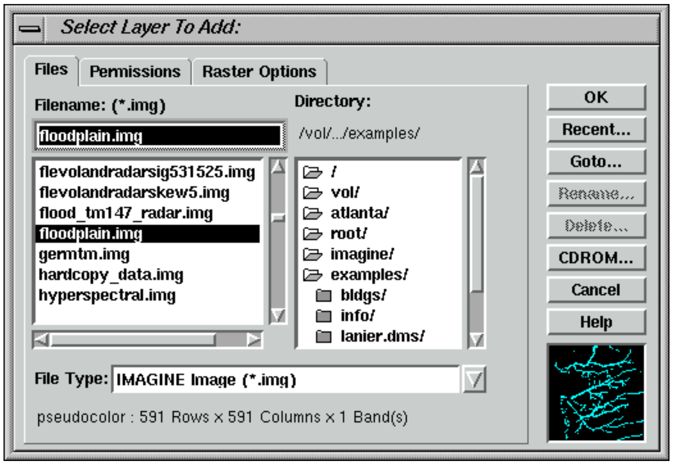
A dialog is a window in which you enter file names, set parameters, and execute processes. In most dialogs, there is very little typing required—simply use the mouse to click on the options you want to use.

Most of the dialogs used throughout the tour guides are reproduced from the software, with arrows showing you where to click. These instructions are for reference only. Follow the numbered steps to actually select dialog options.

For On-Line Help with a particular dialog, click the **Help** button in that dialog.

All of the dialogs that accompany the raster and vector editing tools, as well as the Select Layer To Add dialog, contain a Preview window, which enables you to view the changes you make to the Viewer image before you click **Apply**.

Most of the functions in ERDAS IMAGINE are accessible through dialogs similar to the one below:


Select Layer To Add:
Files
Permissions
Raster Options
flevolandradarsig531525.img
flevolandradarskew5.img
flood_tm147_radar.img
floodplain.img
germtm.img
hardcopy_data.img
hyperspectral.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) floodplain.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

pseudocolor : 591 Rows x 591 Columns x 1 Band(s)

More Information/Help

As you go through the tour guides, or as you work with ERDAS IMAGINE on your own, there are several ways to obtain more information regarding dialogs, tools, or menus, as described below.

On-Line Help

There are two main ways you can access On-Line Help in ERDAS IMAGINE:

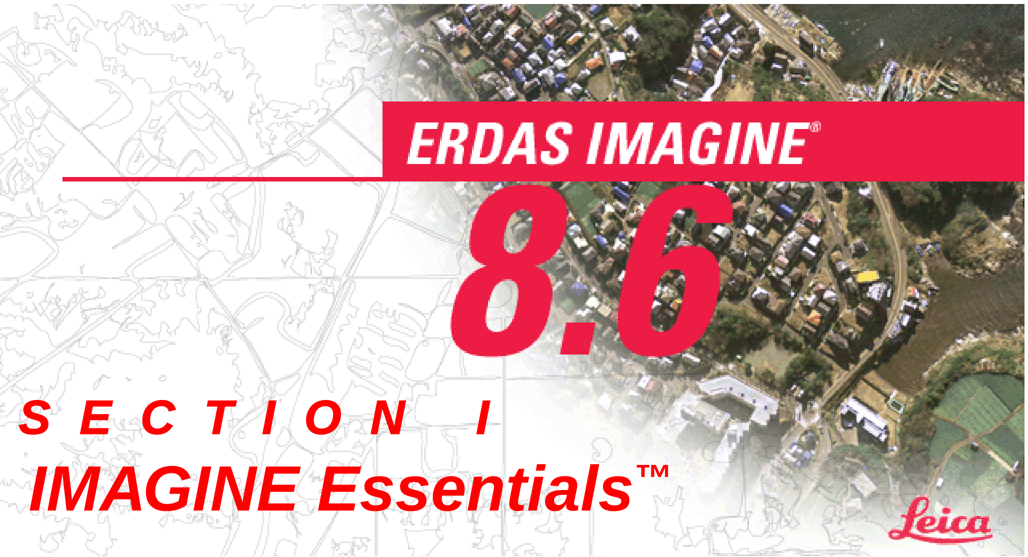
- select the **Help** option from a menu bar
- click the **Help** button on any dialog.

Status Bar Help

The status bar at the bottom of the Viewer displays a quick explanation for buttons when the mouse cursor is placed over the button. It is a good idea to keep an eye on this status bar, since helpful information displays here, even for other dialogs.

Bubble Help

The User Interface and Session category of the Preference Editor enables you to turn on Bubble Help, so that the single-line Help displays directly below your cursor when your cursor rests on a button or frame part. This is helpful if the status bar is obscured by other windows.


ERDAS IMAGINE®
8.6
Leica

SECTION I

IMAGINE Essentials™

CHAPTER ONE

Viewer

Introduction

In this tour guide, you can learn how to:

- set Preferences
- display an image
- query for pixel information
- arrange layers
- adjust image contrast
- link Viewers
- use the Area of Interest (AOI) function
- use the **Raster** menu functions (Raster Attribute Editor, Measurement tools, etc.)

Approximate completion time for this tour guide is 45 minutes.

Display Preferences

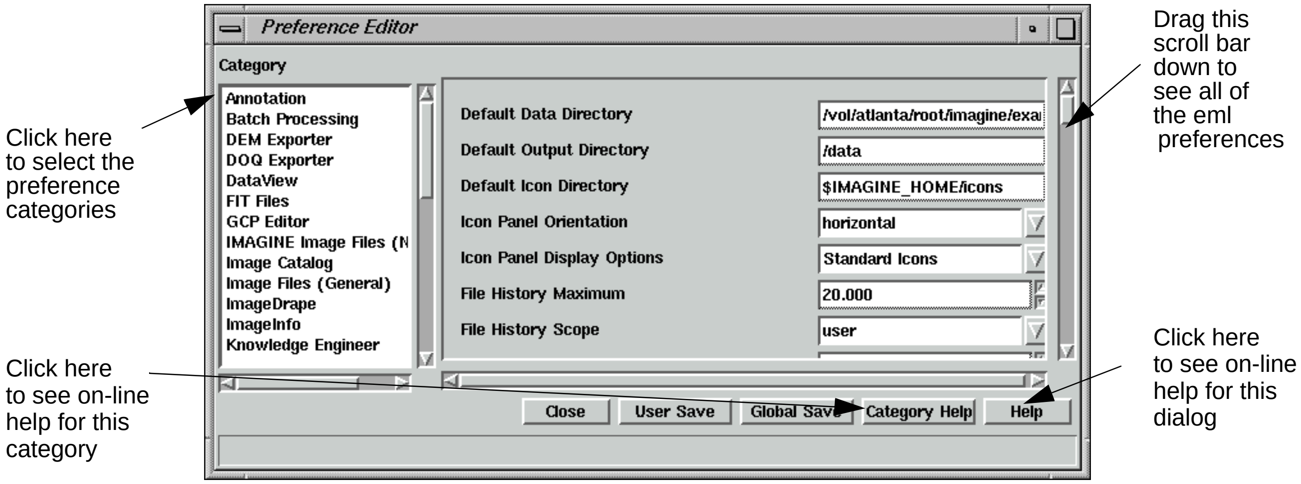
ERDAS IMAGINE allows you to set up default band-to-color gun assignments for Landsat MSS, Landsat TM, SPOT, and AVHRR data in the Preference Editor.

Check Band-to-Color Gun Assignments

ERDAS IMAGINE should be running and a Viewer should be open.

1. Click on the word **Session** in the upper left corner of the ERDAS IMAGINE menu bar.
2. From the **Session** menu, click on **Preferences**.

   The Preference Editor opens.



Preference Editor
Category
Annotation
Batch Processing
DEM Exporter
DOQ Exporter
DataView
FIT Files
GCP Editor
IMAGINE Image Files (N
Image Catalog
Image Files (General)
ImageDrape
ImageInfo
Knowledge Engineer
Close
User Save
Global Save
Category Help
Help

Default Data Directory: /vol/atlanta/root/imagine/exa
Default Output Directory: /data
Default Icon Directory: $IMAGINE_HOME/icons
Icon Panel Orientation: horizontal
Icon Panel Display Options: Standard Icons
File History Maximum: 20.000
File History Scope: user

Click here to select the preference categories

Click here to see on-line help for this category

Drag this scroll bar down to see all of the eml preferences

Click here to see on-line help for this dialog

3. Drag the scroll bar on the right side of the dialog down to see all of the **User Interface & Session** preferences (**User Interface & Session** is the default under **Category**).

   You may change these or any other preferences at any time by selecting the preference category (click on the list below **Category**) and then editing the text in the text entry fields.

4. Under the **User Interface & Session** category in the Preference Editor, locate the preferences for the **AVHRR Band Defaults**, **MSS Band Defaults**, **Spot Band Defaults**, and **TM Band Defaults**.

   The number that is entered for these defaults shows the band that is used for the Red, Green, and Blue color guns in your display. You may change these defaults. These are the band assignments that display in the **Layers to Colors** section of the Select Layer To Add dialog when it opens. These assignments can also be changed in the Select Layer To Add dialog for specific files.

Check Viewer Preferences

1. With the Preference Editor still open, click on the **Category** list and select **Viewer**.

   The Viewer preferences display.

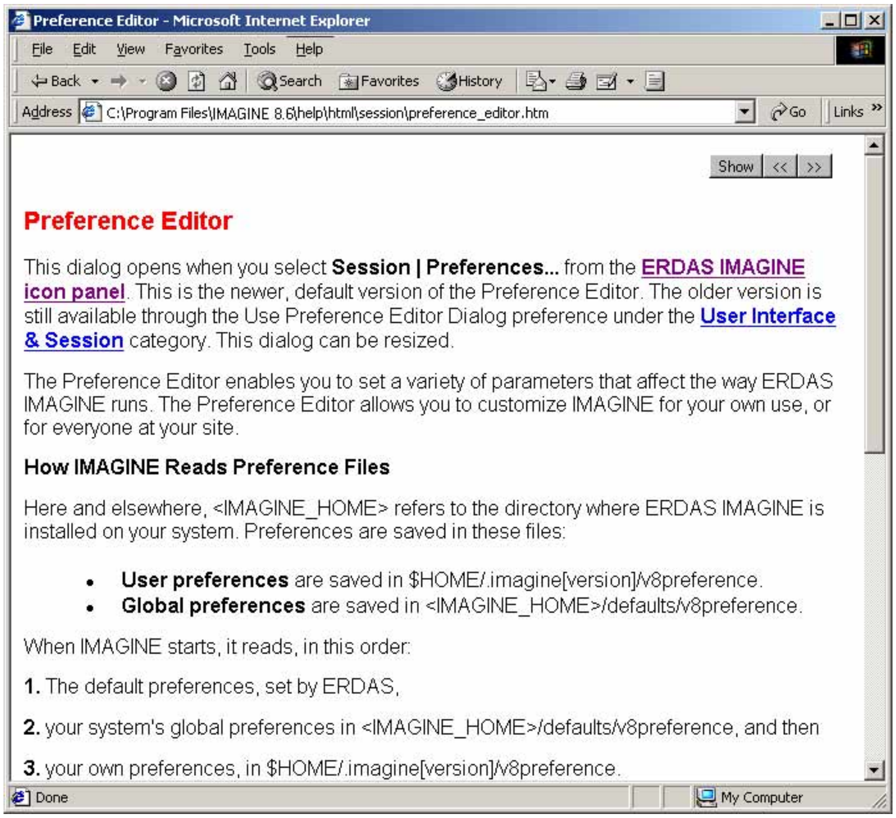
2. Drag the scroll bar on the right of the dialog down to see all of the **Viewer** preferences.

   These preferences control the way the Viewer automatically displays and responds each time it is opened.

Check Preference Editor Help

1. Click on **Help** in the lower right corner of the dialog.

   The On-Line Help for the Preference Editor opens.


Preference Editor - Microsoft Internet Explorer
File Edit View Favorites Tools Help
Back Search Favorites History
Address C:\Program Files\IMAGINE 8.6\help\html\session\preference_editor.htm
Go Links
Show << >>
Done
My Computer

Preference Editor

This dialog opens when you select **Session | Preferences...** from the ERDAS IMAGINE icon panel. This is the newer, default version of the Preference Editor. The older version is still available through the Use Preference Editor Dialog preference under the User Interface & Session category. This dialog can be resized.

The Preference Editor enables you to set a variety of parameters that affect the way ERDAS IMAGINE runs. The Preference Editor allows you to customize IMAGINE for your own use, or for everyone at your site.

How IMAGINE Reads Preference Files

Here and elsewhere, <IMAGINE_HOME> refers to the directory where ERDAS IMAGINE is installed on your system. Preferences are saved in these files:

- **User preferences** are saved in $HOME/.imagine[version]/v8preference.
- **Global preferences** are saved in <IMAGINE_HOME>/defaults/v8preference.

When IMAGINE starts, it reads, in this order:

1. The default preferences, set by ERDAS,

2. your system's global preferences in <IMAGINE_HOME>/defaults/v8preference, and then

3. your own preferences, in $HOME/.imagine[version]/v8preference.

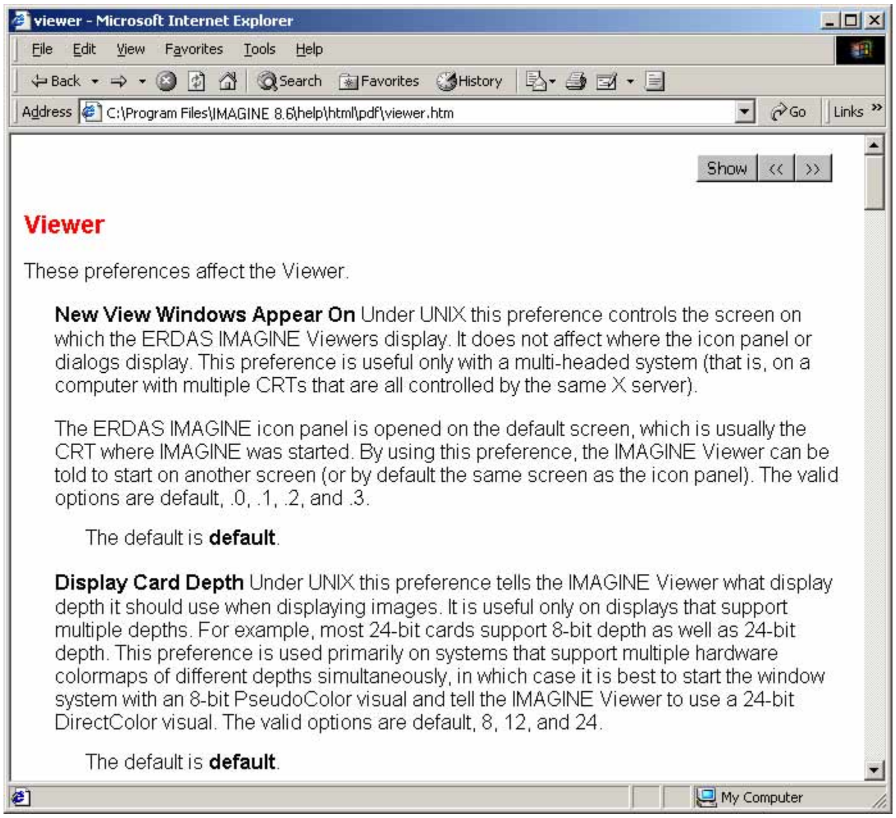
2. When you are through studying the Preference Editor help file, select **File | Exit** from the On-Line Help file menu bar.

   The On-Line Help file closes.

View Category Help

1. Click the **Category Help** button on the Preference Editor.

   The On-Line Help for this category, **Viewer**, opens.


viewer - Microsoft Internet Explorer
File Edit View Favorites Tools Help
Back Search Favorites History
Address C:\Program Files\IMAGINE 8.6\help\html\pdf\viewer.htm
Go Links
Show << >>
My Computer

Viewer

These preferences affect the Viewer.

New View Windows Appear On Under UNIX this preference controls the screen on which the ERDAS IMAGINE Viewers display. It does not affect where the icon panel or dialogs display. This preference is useful only with a multi-headed system (that is, on a computer with multiple CRTs that are all controlled by the same X server).

The ERDAS IMAGINE icon panel is opened on the default screen, which is usually the CRT where IMAGINE was started. By using this preference, the IMAGINE Viewer can be told to start on another screen (or by default the same screen as the icon panel). The valid options are default, .0, .1, .2, and .3.

The default is **default**.

Display Card Depth Under UNIX this preference tells the IMAGINE Viewer what display depth it should use when displaying images. It is useful only on displays that support multiple depths. For example, most 24-bit cards support 8-bit depth as well as 24-bit depth. This preference is used primarily on systems that support multiple hardware colormaps of different depths simultaneously, in which case it is best to start the window system with an 8-bit PseudoColor visual and tell the IMAGINE Viewer to use a 24-bit DirectColor visual. The valid options are default, 8, 12, and 24.

The default is **default**.

2. When you are through studying the Viewer preferences help file, select **File | Exit** from the On-line Help file menu bar.

3. Click the **Close** button on the Preference Editor.

   NOTE: If you have changed any preferences, you can save them at this time by clicking the **User Save** *or* **Global Save** *buttons on the Preference Editor dialog.*

Display an Image

Next, you display a Landsat Thematic Mapper (TM) image of Gainesville, Georgia in a Viewer.

Since the data files in the <IMAGINE_HOME>/examples directory are read only, you may want to copy them to a new directory and change the file permissions. Remember, <IMAGINE_HOME> is the variable name for the directory where ERDAS IMAGINE resides.

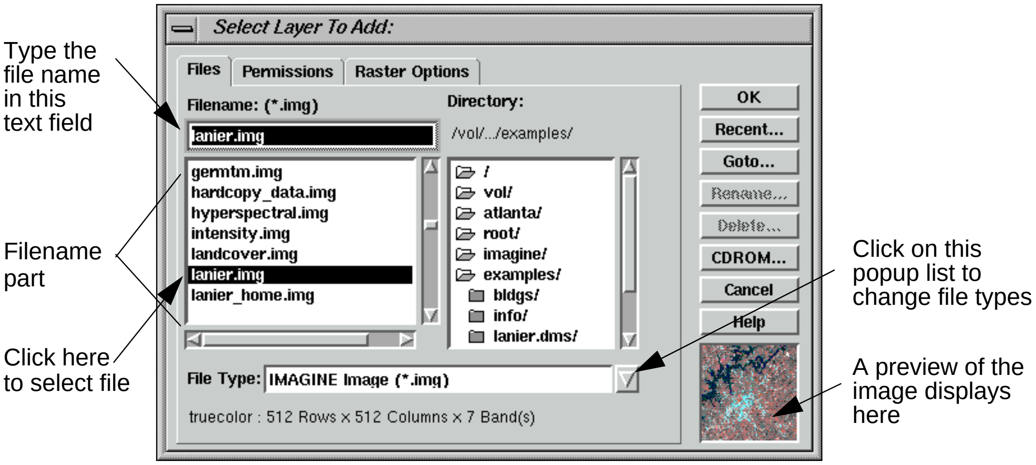
1. In the Viewer menu bar, select **File | Open | Raster Layer**.

You can also open this dialog using either of these two methods:

— use the keyboard shortcut, **Ctrl-r**

— click on this icon in the Viewer tool bar.

The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
germtm.img
hardcopy_data.img
hyperspectral.img
intensity.img
landcover.img
lanier.img
lanier_home.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) lanier.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

truecolor : 512 Rows x 512 Columns x 7 Band(s)

Type the file name in this text field

Filename part

Click here to select file

Click on this popup list to change file types

A preview of the image displays here

2. In the Select Layer To Add dialog, click on the **Recent** button.

   A dialog with a listing of the most recent files you have opened displays. You can individually select these files and then click **OK** to display them quickly in the Select Layer To Add dialog.

3. Click **Cancel** in the List of Recent Filenames dialog.

4. In the Select Layer To Add dialog, click on the **Goto** button.

   A dialog with a listing of the most recent directories you have opened displays. You can individually select these directories, or enter the name of a new directory, and then click **OK** to display that directory quickly in the Select Layer To Add dialog.

5. Click **Cancel** in the Select a Directory dialog.

   NOTE: The **Recent** *and* **Goto** *buttons in the Select Layer To Add dialog are helpful for quickly locating and displaying a file or directory you work with often.*

Filename Part

The framepart under **Filename** is called a filename part. A filename part is a tool used to select specific files for use in an ERDAS IMAGINE function. A filename part consists of:

- a text field—for entering the file name by typing it in, or clicking on, the file from the scroll list.
- a scrolling list—shows the name of all files with the default extension in the selected directory. Files can be selected by clicking on the name in the list.

6. In the filename part of the Select Layer To Add dialog, click on the file **lanier.img**.

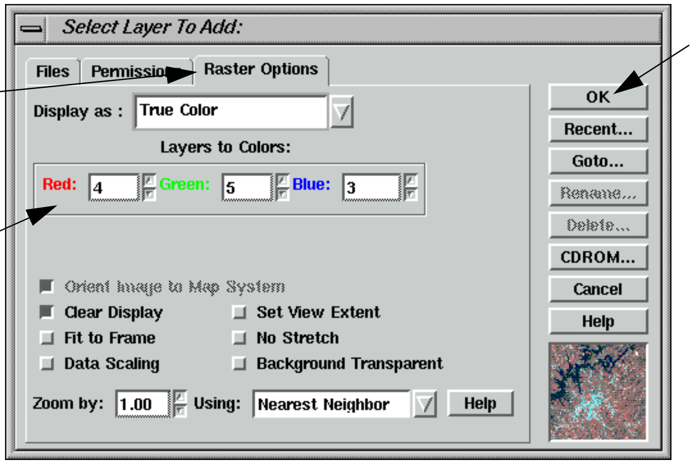
   This is a Landsat TM image of the Gainesville, Georgia area, including Lake Lanier. Information about this file is reported in the bottom, left corner of the Select Layer To Add dialog. This true color image has seven bands, 512 columns, and 512 rows.

7. Click on the **Raster Options** tab at the top of the Select Layer To Add dialog.

   The Raster Options display.

Click this file tab to display the raster options

Change the layer to color band assignment here



Display as : True Color

Layers to Colors:

Red: 4 Green: 5 Blue: 3

☒ Orient Image to Map System

☒ Clear Display

☐ Set View Extent

☐ Fit to Frame

☐ No Stretch

☐ Data Scaling

☐ Background Transparent

Zoom by: 1.00 Using: Nearest Neighbor

Click here to display the image in the Viewer

8. Under **Layers to Colors**, display band **4** in the **Red** color gun, band **5** in the **Green** color gun, and band **3** in the **Blue** color gun.

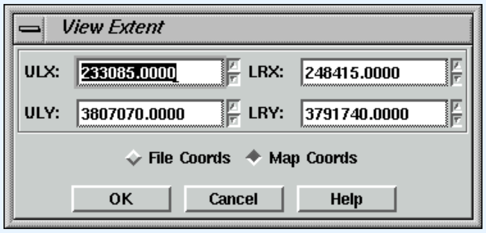
Display Options

1. Note the display options in the Select Layer To Add dialog.

Display Options Defaults

The default settings of the Raster Options tab are briefly described below:

- **Orient Image to Map System**—This checkbox is enabled if calibration is saved to the image file. If there is no calibration, this option is disabled. When enabled, the image is displayed using calibration. Otherwise, the calibration is ignored.
- **Clear Display**—When this checkbox is enabled, and a new image is loaded, the image currently displayed in the Viewer is removed. Disable this checkbox to overlay images.
- **Fit to Frame**—If this checkbox is enabled, the image is magnified or reduced to fit the Viewer window at its current size.
- **Data Scaling**—The Viewer performs a two standard deviation stretch by default. Click this checkbox to select an alternate data range to stretch.
  If you want to save the contrast stretched values with the image, you can use the **Radiometric Enhance | LUT Stretch** option of Image Interpreter.
- **Zoom by**—If **Fit to Frame** is disabled, then you can enter the zoom ratio for the data in this data field.
- **Set View Extent**—Allows you to specify the upper left and lower right coordinates of the portion of the image to display.



ULX: 233085.0000 LRX: 248415.0000
ULY: 3807070.0000 LRY: 3791740.0000
○ File Coords ● Map Coords

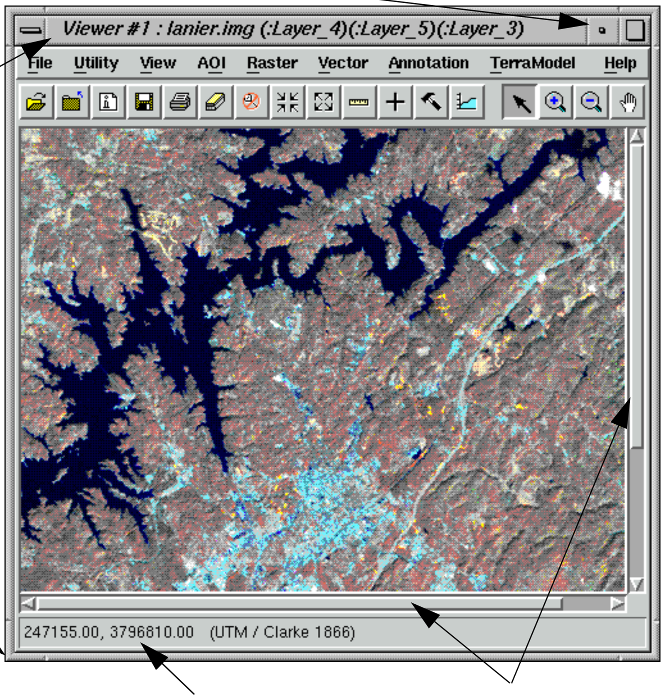
The coordinates in this dialog set the area of the image to display in the Viewer. This is useful if you have an image that is larger than the Viewer window, or if you want only a specific portion of a large image to display in the Viewer.You can also select **View | Scale | Extent** from the Viewer menu bar.

- **No Stretch**—Click to display data without applying the normal two standard deviation stretch.
- **Background Transparent**—Click to make the background of gray scale, pseudocolor, and true color areas transparent—the layer underneath shows through. Background areas are automatically transparent in thematic layers.
- **Using**—Resampling is appropriate if the image is magnified (a magnification factor greater than one). Use one of the following resampling methods: Nearest Neighbor, Bilinear Interpolation, Cubic Convolution, and Bicubic Spline.

These methods of resampling are discussed in "Chapter 9: Rectification" *of the* ERDAS Field Guide.

2. Click **OK** in the Select Layer To Add dialog to display the file.

   The file **lanier.img** is displayed in the Viewer. The name of the file and the layers selected are written in the Viewer title bar.



247155.00, 3796810.00 (UTM / Clarke 1866)

Click here to minimize the window

The title bar shows the Viewer number and name of image displayed

In the title bar, hold and drag to move the Viewer

Drag on any of the corners to resize the Viewer

Single-line help and coordinates display in this Status Bar

Use the scroll bars to roam in an image

Utility Menu Options

The **Utility** menu on the Viewer enables you to access four separate groups of functions:

- inquiry functions
- measurement tool
- layer viewing
- information

Each function group is separated by a line in the pulldown menu.

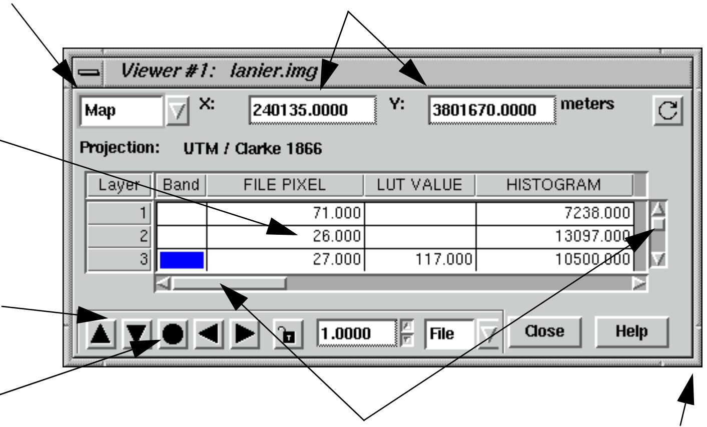
Use Inquiry Functions

You can query a displayed image for information about each pixel using the inquiry functions.

The file **lanier.img** must be displayed in a Viewer.

1. Select **Utility | Inquire Cursor** from the Viewer menu bar.

   A white crosshair displays in the Viewer and the Inquire Cursor dialog opens.



Map projection/spheroid listed here

Drag the crosshair over the image in the Viewer or click here to move it

Pixel information reported in this CellArray

Click arrows to move the Inquire cursor

Click the circle to move the Inquire cursor to the center of the image

Drag the scroll bars for more details

Drag any corner to enlarge the CellArray for more information

Viewer #1: lanier.img

Map X: 240135.0000 Y: 3801670.0000 meters

Projection: UTM / Clarke 1866

Layer	Band	FILE PIXEL	LUT VALUE	HISTOGRAM
1		71.000		7238.000
2		26.000		13097.000
3	🟦	27.000	117.000	10500.000

You can move the Inquire Cursor in the Viewer using any of these methods:

- drag the white crosshair over the image.
- enter new coordinates into the CellArray™ of the Inquire Cursor dialog. The Inquire Cursor moves when you move the mouse cursor back into the Viewer.
- click on the black arrows at the bottom of the Inquire Cursor dialog.

As the crosshair is moved, the information in the Inquire Cursor dialog automatically updates.

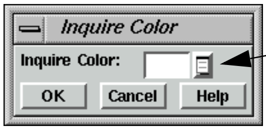
2. The CellArray in the Inquire Cursor dialog reports a variety of pixel information. Drag on the horizontal scroll bar (or enlarge the Inquire Cursor dialog by dragging any corner) to show all of the pixel information available in the CellArray.

Change Inquire Cursor Style

You can change the color and shape of the Inquire Cursor to make it more visible in the Viewer.

1. To change the color of the Inquire Cursor, select **Utility | Inquire Color** from the Viewer menu bar.

   The Inquire Color dialog opens.



Hold on the popup list to select a new color

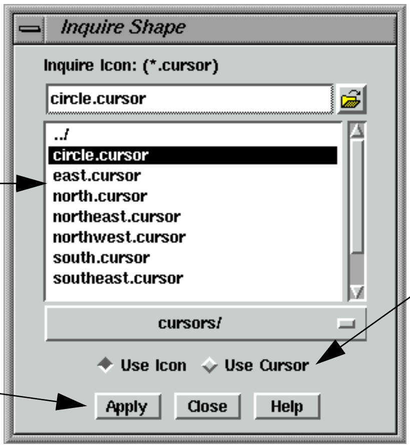
2. Select a new color for the Inquire Cursor by holding on the **Inquire Color** popup list and dragging to select the desired color.

3. Click **OK** in the Inquire Color dialog.

   The Inquire Cursor changes colors.

4. To change the shape of the Inquire Cursor, select **Utility | Inquire Shape** from the Viewer menu bar.

   The Inquire Shape dialog opens


Inquire Shape
Inquire Icon: (*.cursor)
circle.cursor
../
circle.cursor
east.cursor
north.cursor
northeast.cursor
northwest.cursor
south.cursor
southeast.cursor
cursors/
Use Icon
Use Cursor
Apply
Close
Help

Click to select a new shape

Click here to apply the new shape to the Inquire Cursor

Click to make the Inquire Cursor an icon or a cursor

5. Click on **circle.cursor** in the scroll list that displays, then click **Apply**.

   The Inquire Cursor becomes a circle.

6. In the Inquire Shape dialog, click on the **Use Cursor** button, then **Apply** to return the Inquire Cursor to the original crosshair shape.

7. Click **Close** in the Inquire Shape and the Inquire Cursor dialogs.

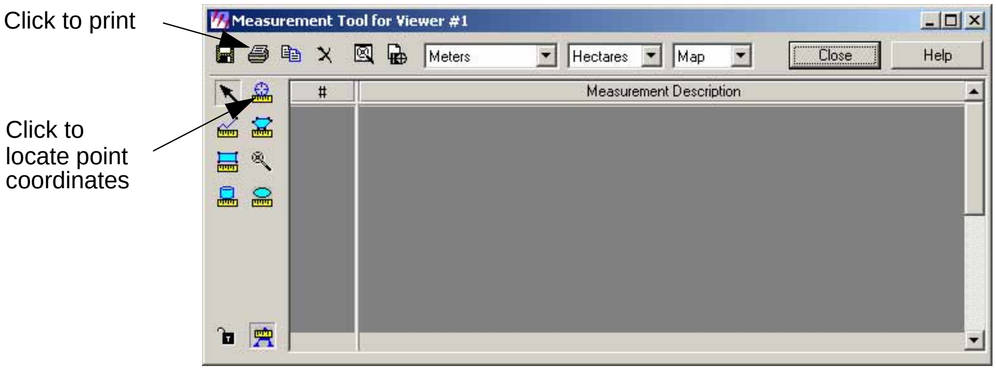
   The Inquire Cursor is cleared from the Viewer.

Take Measurements

The Measurement tool enables you to measure points, lines, polygons, rectangles and ellipses in the displayed layer. Both distance and area are reported in the units you select.

1. Click on the Measurement icon in the Viewer tool bar or select **Utility | Measure** from the Viewer menu bar.

   The Measurement Tool viewer opens.


Click to print

Click to locate point coordinates

2. Click on the Measure Positions icon in the Measurement tool bar. This tool gives the individual point coordinates (x, y) in the image.

3. Move the cursor into the Viewer and click anywhere.

   In the Measurement Tool viewer, the location of the point displays in the type of units in which the file is saved. You may select different display units from the popup lists in the top toolbar.

4. Next, click on the Polyline icon in the Measurement Tool viewer tool bar.

5. Move the cursor into the Viewer and click once at the beginning of a line feature then drag the mouse to extend the line along the feature. Click to add a vertex at each point. Middle-click (or double-click, depending on how your Preferences are set) to end the measurement.

   The length is displayed in the Measurement Tool CellArray.

The Measurement Tool

The Measurement Tool can create a new annotation layer on top of your image. Simply click the Annotation tool and a new layer is automatically created. While this tool is enabled, the measurement features (points, polylines, polygons, rectangles, ellipses, etc.) are added to the annotation layer as well as a text box containing the measured values. Click the tool again to turn this feature off.

The annotation layer may be saved and used with other images with the same geographic area.

These annotation objects may be moved and resized, but the measured values in the text boxes are not updated.

6. Click the print icon to print and a Print dialog opens, which allows you to enter or select the printer to be used.

7. Select the **Printer** and click **Print** (or **OK**) in the Print dialog. If you do not wish to print, click **Cancel**.

8. Experiment with the other measurement tools if you like, and when you are done, click the **Close** button in the top toolbar.

   You are asked if you want to save the measurements. Save them if you like. You can click the Save icon at any time to save your measurements.

Click the **Help** *button to view the on-line help for the measurement tools.*

View Menu Options

Arrange Layers

ERDAS IMAGINE should be running, and **lanier.img** should be displayed in a Viewer.

1. In the Viewer tool bar, click the Open icon to open another layer on top of **lanier.img**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, click on **lnsoils.img**. This is a thematic soils file of the Gainesville, Georgia area.

3. Click on the **Raster Options** tab at the top of the Select Layer To Add dialog.

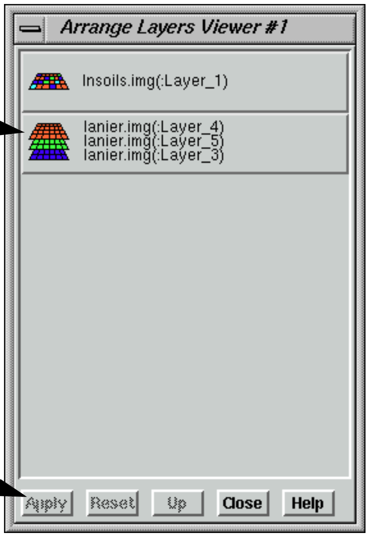
4. Check to be sure that the **Clear Display** checkbox is disabled (not selected), so that **lanier.img** is not cleared from the Viewer when **lnsoils.img** is displayed.

5. Click **OK** in the Select Layer To Add dialog to display the file.

   Now, both **lanier.img** and **lnsoils.img** are displayed in the same Viewer, with **lnsoils.img** on top.

6. To bring **lanier.img** to the top of the Viewer, select **View | Arrange Layers** from the Viewer menu bar.

   The Arrange Layers dialog opens.



Click and drag this box to the top to change the order of the displayed layers

Click here to redisplay layers in the new order

7. In the Arrange Layers dialog, drag the **lanier.img** box above the **lnsoils.img** box, as illustrated above.

   When you release the mouse button, the layers are rearranged in the Arrange Layers dialog so that the **lanier.img** box is first.

8. Click **Apply** in the Arrange Layers dialog to redisplay the layers in their new order in the Viewer.

   The layers are now reversed.

9. Click **Close** in the Arrange Layers dialog.

Zoom

In this section, you zoom in by a factor of 2 and create a magnifier window. Once the image is enlarged, you can roam through it.

lanier.img should be displayed on top of **lnsoils.img** in a Viewer at a magnification of 1 (this is the case if you have been following through this tour guide from the beginning).

1. Select **View | Zoom | In by 2** from the Viewer menu bar.

   The images are redisplayed at a magnification factor of 2.

   The Zoom options are also available from:

   — *the* **Quick View** *menu (*right-hold on the Viewer image) under**Zoom | Zoom In by 2**

   — *the Viewer tool bar by clicking this icon.*

2. Move the scroll bars on the bottom and side of the Viewer window to view other parts of the image.

   To move by small increments, you can click the small triangles at either end of the scroll bars. To move by larger increments, drag the scroll bars.

   You can also enlarge the Viewer window by dragging any corner.

3. Select **View | Create Magnifier** from the Viewer menu bar.

   A white cursor box opens in the center of the image. This area is displayed in a small magnifier window that opens over the top corner of the Viewer.

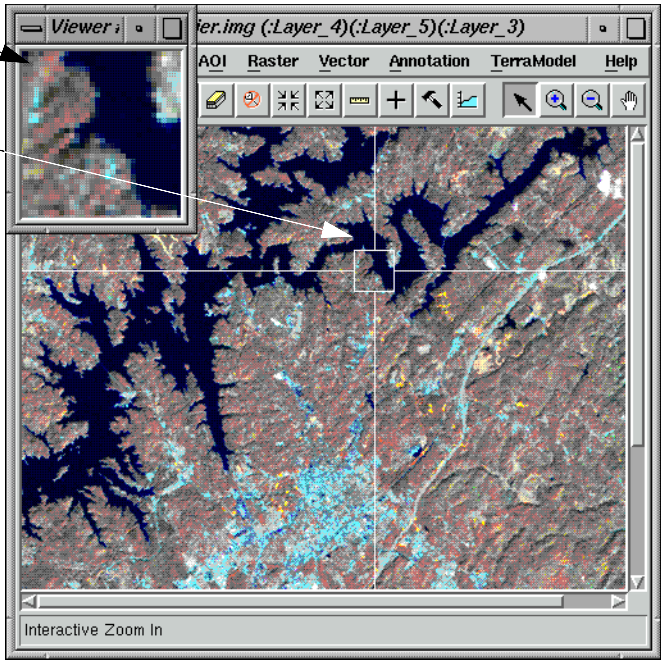


Image in this new Viewer is a magnified view of the image area under this cursor box

4. With your pointer inside the white cursor box, hold and drag the box around the image.

   The data in the magnifier window change as the cursor box is moved over the image. This technique is called chip extraction, which is used in the Rectification tools to help you precisely identify ground control points (GCPs).

5. In the Viewer menu bar, select **File | Close Other Viewers** to close the magnifier window.

Magnifying Areas

There are four ways to change the size of the area magnified:

- With the cursor on any corner (or side) of the cursor box, drag the box until it is the desired size.
- Place the cursor on the lower right corner of the magnifier window and drag the magnifier window until it is the desired size.
- Press the space bar to enter precise positioning coordinates in the inquire box dialog.
- Use the **Quick View** menu (from the right mouse button) or the **View** menu (from the Viewer menu bar) to zoom in either Viewer.

As you try these methods, you notice that each change in size is reflected in the other window. As the cursor box is adjusted, the magnification in the magnifier window is adjusted to accommodate the new area. Likewise, as the magnifier window is adjusted, the cursor box changes to reflect the new size and proportion.

Other methods of zooming in and out of imagery are Animated Zoom, Box Zoom, and Real-time Zoom.

Animated Zoom

Animated Zoom enables you to zoom in and out of the Viewer's image in a series of steps that are similar to animation. The image is resampled after it is magnified or reduced.

Display **lanier.img** in the Viewer.

1. Select **Session | Preferences**.
2. In the Preference Editor dialog, select **Viewer** from the **Category** list.
3. Click the checkbox for **Enable Animated Zoom**.
4. Click **User Save** then **Close** in dialog, and go back to the Viewer.
5. Click the Zoom In By Two icon.

The Viewer zooms into the image in a simulated animation by a factor of 2. The Viewer center is maintained.

6. Click the Zoom Out By Two icon.

The Viewer zooms out of the image in a simulated animation by a factor of 2. The Viewer center is maintained.

7. Click either the Interactive Zoom In or the Interactive Zoom Out icon.

8. Click on a location on the image.

The Viewer recenters the image to that location and zooms in or out in a simulated animation by a factor of 2.

Animated zoom also works with **View | Zoom | In by X** *and* **Out by X**.

Box Zoom

Box Zoom is used to select a boxed area in the image. When zooming in or out by using the zoom recentering icons, the boxed image enlarges or reduces within the Viewer.

Display **lanier.img** in the Viewer.

1. Select **Session | Preferences**.
2. In the Preference Editor dialog, select **Viewer** from the **Category** list.
3. Click to select **Enable Box Zoom**.
4. Click **User Save** then **Close** in dialog, and go back to the Viewer.
5. Click the Interactive Zoom In icon.
6. Click and drag a box in the image.

   The selected area of the image is magnified to fit the Viewer.
7. Select the Interactive Zoom Out icon.
8. Click and drag a box in the image.

   The area displayed in the Viewer is reduced to fit in the box. Space surrounding the reduced image is populated with available imagery.

Real-time Zoom

When you select either of the Interactive Zoom tools, you can to zoom into and out of images in real time by holding the middle mouse button and moving the mouse upward and downward over the image.

NOTE: You can also hold down the Control key and press on the left mouse button to zoom in real time.

Display **lanier.img** in the Viewer. There is no need to set up a preference for this feature.

1. Select either of the Interactive Zoom icons.
2. Position the cursor in the Viewer, and hold the middle mouse button.
3. Move the mouse forward to zoom in on the image.

   The image magnifies at a constant rate, depending on how far forward you move the mouse.
4. Hold the middle mouse button and move the mouse backward.

   The image reduces at a constant rate, depending on how far downward on the image you move the mouse.

Viewer

Display Two Images

Two or more Viewers can be geographically or spectrally linked so that when you roam in one image, that area is simultaneously displayed in the linked Viewer(s).

Types of Linking

- Geographically linked—the same image area displays in all linked Viewers.
- Spectrally linked—enhancements made to an image are also made in other Viewers if that same image, or portions of it, are displayed in other Viewers.

lanier.img should be displayed on top of **lnsoils.img** in a Viewer window, at a magnification of 2.

1. Drag on a lower corner of the Viewer so that it occupies the entire left half of the screen.
2. In the Viewer menu bar, select **View | Split | Split Horizontal**.

   The Viewer is divided in half, horizontally, to form two Viewers.

3. In the tool bar of the new Viewer, click the Open icon.

   The Select Layer To Add dialog opens.

4. In the Select Layer To Add dialog under **Filename**, click on the file **lnsoils.img**.
5. Click on the **Raster Options** tab at the top of the dialog.
6. Confirm that **Zoom by** is set to **1.00**.
7. Click **OK** in the Select Layer To Add dialog.

   The file **lnsoils.img** is displayed in the second Viewer.

Link Viewers

1. In the first Viewer, select **View | Link/Unlink Viewers | Geographical**.

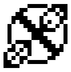
   The Link/Unlink Instructions display.

2. Move your pointer to the second Viewer.

   The pointer becomes a Link symbol.

3. Move the pointer to the first Viewer.

The No Link symbol displays as the cursor in the first Viewer. Clicking in this Viewer discontinues the link operation.



4. To link the Viewers, click anywhere in the second Viewer.

   The two Viewers are now linked. A white cursor box opens over the image in the second Viewer, indicating the image area displayed in the first Viewer.

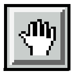
   You can move and resize this cursor box as desired, and the image area in the first Viewer reflects each change. This is similar to the magnification box you used earlier.

Compare Images

1. Drag the cursor box in the second Viewer to a new location. The image area selected in the second Viewer is displayed in the first Viewer.
2. Drag the scroll bars in the first Viewer to roam in the image.

   The white cursor box in the second Viewer moves as the image area in the first Viewer changes.

You could also use the Roam icon in the Viewer tool bar to roam over the image. Just move the hand across the Viewer image to change the view.



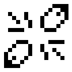
Unlink Viewers

1. In either Viewer, select **View | Link/Unlink Viewers | Geographical** to unlink the Viewers.

   The Link/Unlink Instructions display.

2. Move the pointer to the other Viewer.

   The unlink cursor displays.



3. Click anywhere inside the Viewer to unlink the Viewers.
4. In the menu bar of the second Viewer, select **File | Close**.

   The second Viewer closes.

5. In the first Viewer, select **File | Clear** to clear the Viewer.

Raster Menu Options

Create an AOI Layer

These options allow you to define an AOI in the image, excluding other parts of the image. Specific processes can be applied to this AOI only, which can save considerable time and disk space. The option to use a specified AOI for processing is available from many dialogs throughout ERDAS IMAGINE.

This exercise tells you how to create an AOI layer that can be saved as a file and recalled for later use.

NOTE: Each Viewer can display only one AOI layer at a time.

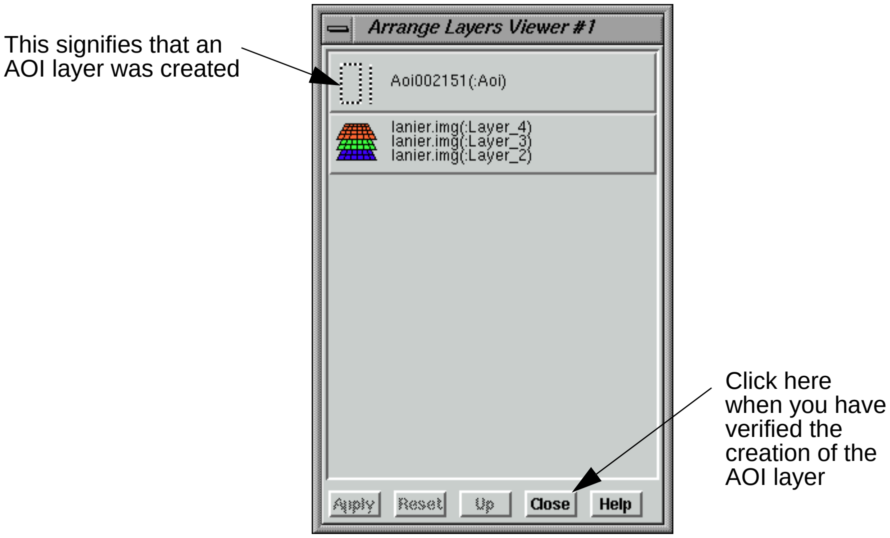
Display **lanier.img** in a Viewer. You must have an image displayed in the Viewer to create an AOI layer.

1. Select **File | New | AOI Layer** from the Viewer menu bar.

   ERDAS IMAGINE creates an AOI layer.

2. Select **View | Arrange Layers** from the Viewer menu bar to verify that the AOI layer has been created.

   The Arrange Layers dialog opens, and should look similar to the following example:


This signifies that an AOI layer was created
Arrange Layers Viewer #1
Aoi002151(:Aoi)
lanier.img(:Layer_4)
lanier.img(:Layer_3)
lanier.img(:Layer_2)
Apply
Reset
Up
Close
Help
Click here when you have verified the creation of the AOI layer

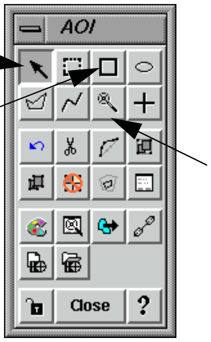
3. After verifying the creation of the AOI layer, click **Close** in the Arrange Layers dialog.

   Later, you are asked to name the layer and save it to a file.

Open AOI Tools

1. Select **AOI | Tools** from the Viewer menu bar (or click the Tools icon on the tool bar).

   The AOI tool palette displays.



Click to select an AOI

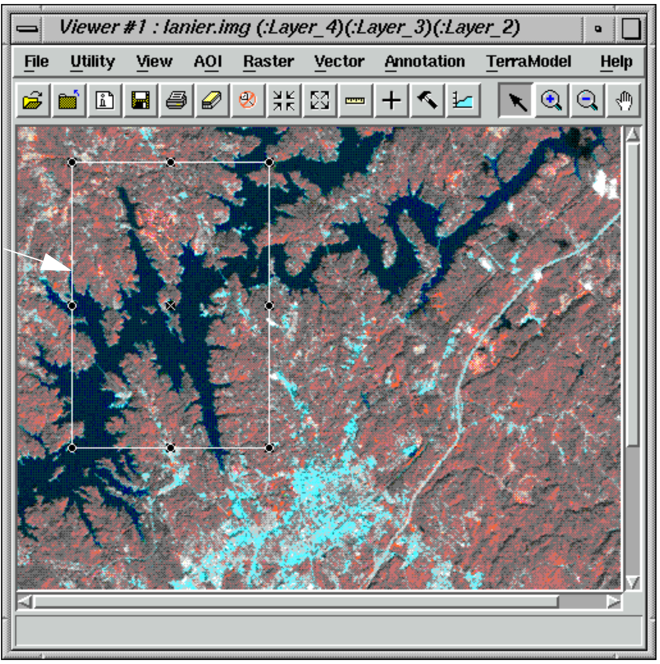
Click to draw a rectangular AOI

Click to plant a seed to grow a regional AOI

2. Click on the Rectangle icon in the AOI tool palette.

3. Move the cursor into the Viewer window. Drag and then release to draw a rectangle over the AOI. Include a portion of the water when designating the AOI.

   A rectangular AOI displays in the Viewer.


Viewer #1 : lanier.img (:Layer_4)(:Layer_3)(:Layer_2)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Rectangular AOI

Selecting AOIs

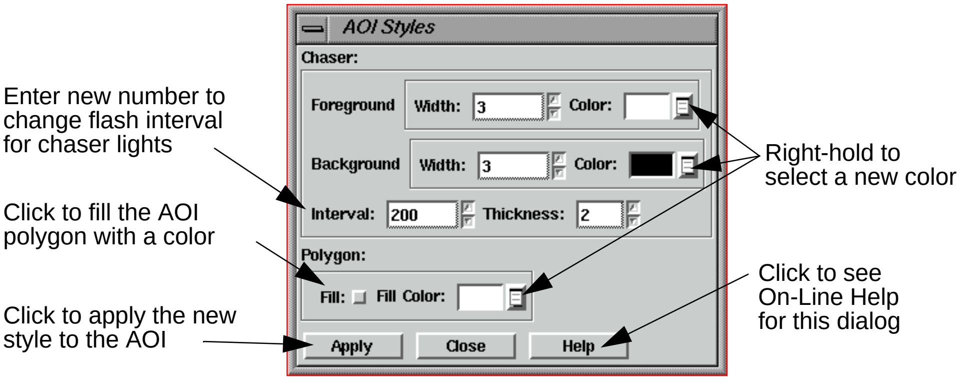
Following are some tips regarding the selection of the AOI:

- You can move the AOI by dragging the AOI to a new location.
- You can resize the AOI by dragging any of the handles at the corners and sides of the bounding box, or by pressing the space bar to enter precise coordinates.
- The **x** in the center of the bounding box marks the center coordinate of the AOI.

Select Styles

1. Select **AOI | Styles** from the Viewer menu bar.

   The AOI Styles dialog opens.



AOI Styles

Chaser:

Foreground Width: 3 Color:

Background Width: 3 Color:

Interval: 200 Thickness: 2

Polygon:

Fill: ☐ Fill Color:

Enter new number to change flash interval for chaser lights

Click to fill the AOI polygon with a color

Click to apply the new style to the AOI

Right-hold to select a new color

Click to see On-Line Help for this dialog

   This dialog enables you to change the style of the AOI display.

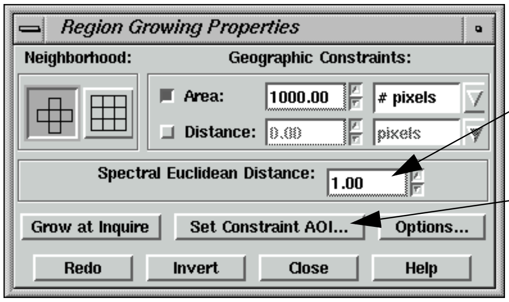
2. Experiment in the AOI Styles dialog with the line widths and colors to find a style that looks best on the displayed image.

3. When you are finished, click **Close** in the AOI Styles dialog.

Set Seed Properties

1. Next, select **AOI | Seed Properties** from the Viewer menu bar.

   The Region Growing Properties dialog opens.



■ Area: 1000.00 # pixels

☐ Distance: 0.00 pixels

Spectral Euclidean Distance: 1.00

Adjust the Spectral Euclidean Distance here

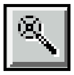
Click to select an AOI to constrain the region growing

This dialog enables you to define the region that grows from the seed.

2. In the Region Growing Properties dialog, change the **Spectral Euclidean Distance** to **5.00**.
3. Click **Set Constraint AOI** in the Region Growing Properties dialog.

   The Choose AOI dialog opens.

4. In the Choose AOI dialog, select **Viewer** under **AOI Source** and then click **OK**.
5. Click the Region Grow AOI icon in the AOI tool palette.



   Click this tool to plant seeds, or points in the Viewer, from which to grow a regional AOI. The region grows in the Viewer as an AOI that can be selected.

6. Move the cursor into the Viewer window and click on the water inside the rectangular AOI to indicate where you want the region growing to take place.

   A status meter displays in the status bar of the Viewer. You may click **Cancel** to terminate the region grow process. The meter dismisses when the region growing process is complete. The area you selected in the Viewer is surrounded by a second bounding box and chaser lights.

7. Click **Close** in the Region Growing Properties dialog.

Save AOI

1. Select **File | Save | AOI Layer As** from the Viewer menu bar.

   The Save AOI As dialog opens. This dialog allows you to save the selected AOIs as a layer (.aoi extension) that can be used again for other functions.

2. Enter a name for the AOI layer under **Save AOI as** (the .aoi extension is added automatically). Pay special attention to the directory where the file is saved, so you can find the layer later.

If you wanted to save specific AOIs only, you could turn on the **Selected Only** *checkbox in the Save AOI As dialog, and only selected AOIs would be saved to a file.*

3. Click **OK** in the Save AOI As dialog.

   This layer can now be used in any dialog where a function can be applied to a specific AOI layer. You can also edit this layer at any time, adding or deleting areas.

Arrange Layers

1. Select **View | Arrange Layers** from the Viewer menu bar.

   The Arrange Layers dialog opens.

2. In the Arrange Layers dialog, right-hold over the AOI Layer and select **Delete Layer** from the **AOI Options** menu.

3. Click **Apply** and then **Close** in the Arrange Layers dialog.

   The file **lanier.img** is redisplayed in the Viewer without the AOI layer.

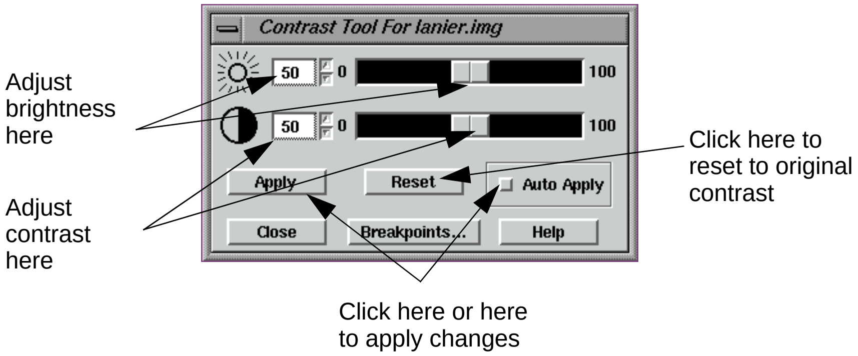
Adjust Image Contrast

When images are displayed in ERDAS IMAGINE, a linear contrast stretch is applied to the data file values, but you can further enhance the image using a variety of techniques.

The file **lanier.img** should be displayed in a Viewer.

1. In the Viewer menu bar, select **Raster | Contrast | Brightness/Contrast**.

   The Contrast Tool dialog opens.


Contrast Tool For lanier.img
50
0
100
50
0
100
Apply
Reset
☐ Auto Apply
Close
Breakpoints...
Help
Adjust brightness here
Adjust contrast here
Click here to reset to original contrast
Click here or here to apply changes

2. In the Contrast Tool dialog, change the numbers and/or use the slider bars to adjust the image brightness and contrast.

3. Click **Apply**.

   The image in the Viewer is redisplayed with new brightness values.

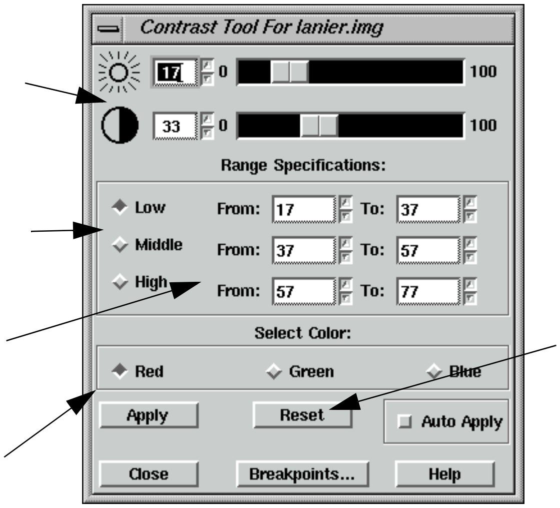
4. Click **Reset** and **Apply** in the Contrast Tool dialog to undo any changes made to the Viewer image.

5. Click **Close** in the Contrast Tool dialog.

Use Piecewise Linear Stretches

1. In the Viewer menu bar, select **Raster | Contrast | Piecewise Contrast**.

   The Contrast Tool dialog for piecewise contrast opens.



17

33

◆ Low From: 17 To: 37

◇ Middle From: 37 To: 57

◇ High From: 57 To: 77

◆ Red ◇ Green ◇ Blue

☐ Auto Apply

Adjust contrast and brightness here

Specify range of lookup table to modify here

Set lookup table ranges here

Select color gun to affect contrast

Click here to reset to the original lookup table

The Contrast Tool

This tool enables you to enhance a particular portion of an image by dividing the lookup table into three sections: low, middle, and high. You can enhance the contrast or brightness of any section using a single color gun at a time. This technique is very useful for enhancing image areas in shadow, or other areas of low contrast.

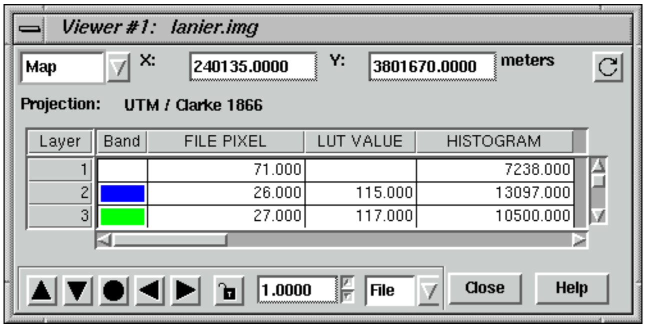
The brightness value for each range represents the midpoint of the total range of brightness values occupied by that range.

The contrast value for each range represents the percent of the available output range that particular range occupies.

As one slider bar is moved, the other is automatically adjusted, so that there is no gap in the lookup table. This tool is set up so that there are always pixels in each data file value from 0 to 255. You can manipulate the percentage of pixels in a particular range, but you cannot eliminate a range of data file values.

2. With your pointer over the image in the Viewer, right-hold **Quick View | Inquire Cursor**.

   The Inquire Cursor dialog opens and an Inquire Cursor is placed in the Viewer.



Map X: 240135.0000 Y: 3801670.0000 meters

Projection: UTM / Clarke 1866

Layer	Band	FILE PIXEL	LUT VALUE	HISTOGRAM
1		71.000		7238.000
2	🟦	26.000	115.000	13097.000
3	🟩	27.000	117.000	10500.000

1.0000 File

3. In the Viewer, drag the intersection of the Inquire Cursor to the lake. Move the Inquire Cursor over the water while keeping an eye on the lookup table values in the blue color gun, as reported in the Inquire Cursor dialog.

This gives you an idea of the range of data file values in the water. You can stretch this range to bring out more detail in the water.

4. In the Contrast Tool dialog, click **Blue** under **Select Color**.

5. Under **Range Specifications**, set the **Low** range **From 34 To 55** and press Return on your keyboard.

6. Drag the Brightness slider bar (the top slider bar) to **50**.

7. Click **Apply** in the Contrast Tool dialog.

The water now has more contrast and shows more detail.

If your image is at a magnification of 1, this new detail may be difficult to see. You can zoom in to a magnification of 2 using the **Quick View** menu in the Viewer.

8. In the Contrast Tool dialog, click **Reset** and then **Apply** to return the image to the original lookup table values.

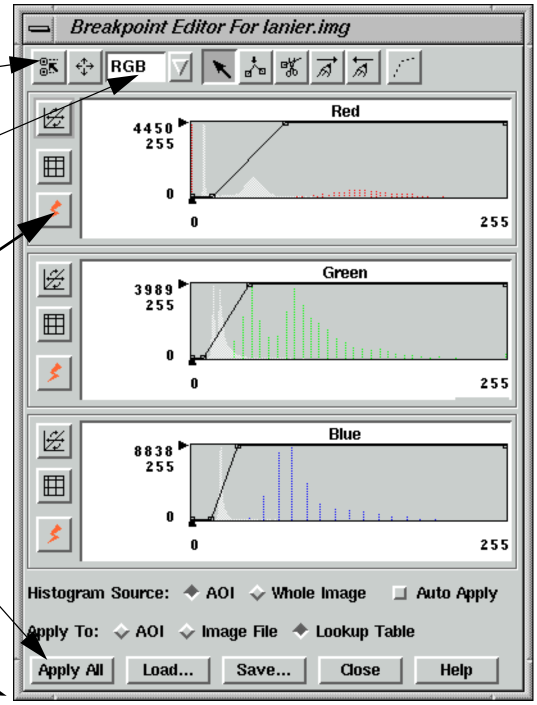
9. Click **Close** in the Contrast Tool dialog.

10. Click **Close** in the Inquire Cursor dialog.

Manipulate Histogram

1. In the Viewer menu bar, select **Raster | Contrast | Breakpoints**.

The Breakpoint Editor opens.


Breakpoint Editor For lanier.img
RGB
Red
4450
255
0
0
255
Green
3989
255
0
0
255
Blue
8838
255
0
0
255
Apply All
Load...
Save...
Close
Help

Histogram Source: ◆ AOI ◇ Whole Image ☐ Auto Apply

Apply To: ◇ AOI ◇ Image File ◆ Lookup Table

Allows you to set histogram display options

Indicate which histograms to view

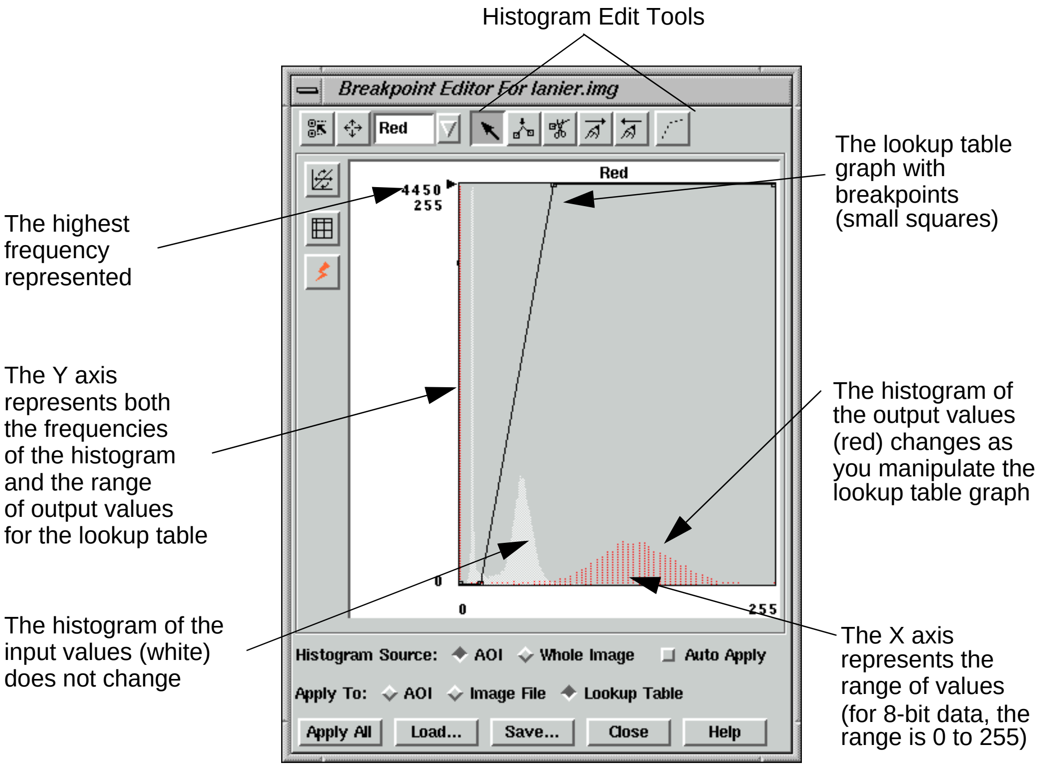
Click to apply changes to the red histogram only

Click to apply changes made to all histograms

Enlarge the histograms by dragging on any corner

2. Click on the popup list at the top of the Breakpoint Editor and select **Red**.

Each of the three histogram graphics in the Breakpoint Editor can be expanded up to full size by selecting the appropriate histogram from the popup menu at the top of the Break Point Editor. The parts of the histogram graphic are described in the following illustration.



Histogram Source: ◆ AOI ◇ Whole Image ☐ Auto Apply

Apply To: ◇ AOI ◇ Image File ◆ Lookup Table

The highest frequency represented

The Y axis represents both the frequencies of the histogram and the range of output values for the lookup table

The histogram of the input values (white) does not change

The lookup table graph with breakpoints (small squares)

The histogram of the output values (red) changes as you manipulate the lookup table graph

The X axis represents the range of values (for 8-bit data, the range is 0 to 255)

3. Click on the popup list at the top of the Breakpoint Editor and select **RGB**.

   All three histograms redisplay in the Breakpoint Editor.

4. Experiment by dragging the breakpoints of the lookup table graphs in the different color guns (**Red**, **Green**, and **Blue**).

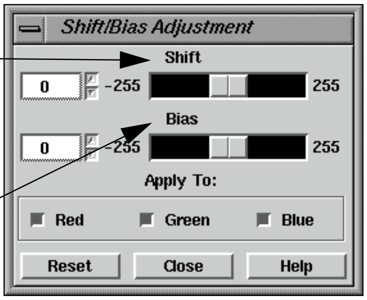
5. Click **Apply All** in the Breakpoint Editor to view the results of your changes in the image.

6. To undo the edits you just made, select **Raster | Undo** from the Viewer menu bar.

Adjust Shift/Bias

1. In the Breakpoint Editor, click the Shift/Bias icon on the tool bar.

   The Shift/Bias Adjustment dialog opens.



Shift moves the lookup table graph left and right (X direction)

Bias moves the lookup table graph up and down (Y direction)

The lookup table graph and the output histogram are updated in the Histogram Tool dialog as you manipulate the information in the Shift/Bias Adjustment dialog.

2. In the Shift/Bias Adjustment dialog, drag the **Shift** slider bar to the right.

   Notice that the value in the number field to the left increases as you move the slider bar. This is the number of pixels that the lookup table graph is moved.

3. In the Shift/Bias Adjustment dialog, double-click on the number in the **Shift** number field and change the number field to **20**. Press Enter on your keyboard.

4. In the Breakpoint Editor, click **Apply All**.

   The image is redisplayed using the new lookup table. It is very dark.

5. In the Shift/Bias Adjustment dialog, return the **Shift** value to **0**.

6. In the Breakpoint Editor, click **Apply All** to return the image to its original contrast.

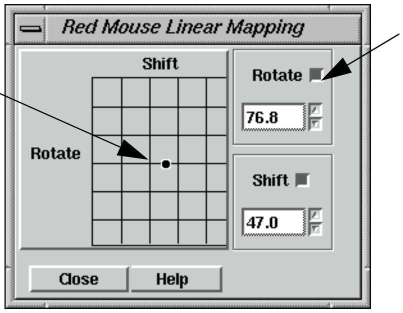
7. Repeat steps (2. through step 6., using the **Bias** option.

8. When you are finished, click **Close** in the Shift/Bias Adjustment dialog.

Use Mouse Linear Mapping

1. In the Breakpoint Editor, click the Red Mouse Linear Mapping icon, which is located on the left border of the **Red** histogram.

   The Red Mouse Linear Mapping dialog opens.



Rotate ☐ 76.8

Shift ☐ 47.0

Move this dot left and right to shift the lookup table graph. Move the dot up and down to rotate the lookup table graph.

Click here to disable this option

2. In the Red Mouse Linear Mapping dialog, click the **Rotate** button to disable the rotate option.
3. Drag the dot in the center of the grid left or right to shift the red lookup table graph.
4. In the Breakpoint Editor, click the Run icon for the red histogram to update the image in the Viewer.
5. In the Red Mouse Linear Mapping dialog, click the **Rotate** button to turn it on and the **Shift** button to turn it off.
6. Drag the dot in the grid up or down to change the slope of the lookup table graph.
7. Click the Run icon for the red histogram to update the Viewer image.
8. Click **Close** in the Red Mouse Linear Mapping dialog.
9. With your cursor over the red histogram graph, right-hold **Graph Options | Undo All Edits**.
10. Click **Apply All** in the Breakpoint Editor to return the Viewer image to its original contrast.
11. Click **Close** in the Breakpoint Editor.
12. Select **File | Clear** from the Viewer menu bar.

Linear Mapping

Moving the dot in the center of the grid left and right shifts the lookup table graph in the histogram graphic left or right. Moving the dot up and down rotates the lookup table graph, changing the slope. Up rotates the graph counterclockwise and down rotates the graph clockwise.

As the dot is moved, the numbers on the right side of the dialog are automatically updated. The **Rotate** number reports the angle of the rotation ramp, with 180 being a straight horizontal line and 90 being a straight vertical line. The **Shift** number reports the pixel value at the center of the lookup table graph.

Raster Editor

The Raster Editor enables you to edit portions of the displayed image using various tools in the Viewer **Raster** menu. When a specific raster editing tool is in use, that tool locks the Viewer, therefore, work with one tool must be completed before opening another one.

All of the dialogs that accompany the raster editing tools contain a preview window, which enables you to view the changes you make to the Viewer image before you click **Apply**.

Prepare (UNIX)

You must have a writable file displayed to use this function. Follow the steps below to create a writable file to work with.

1. In a command window, copy **lndem.img** to **testdem.img** by typing the following:

   ```
   cp   $IMAGINE_HOME/examples/lndem.img   <your directory path>/testdem.img
   ```

   Press Return on your keyboard.

2. Change read/write permissions by typing the following in the command window:

   ```
   chmod   644   testdem.img
   ```

   Press Return on your keyboard and close the command window.

Prepare (PC)

1. Open the Explorer.
2. Copy **lndem.img** from the <IMAGINE_HOME>/examples directory to the directory of your choice.
3. Right-click and select **Rename** to rename the file **testdem.img**.
4. Right-click on the file, and select **Properties**.
5. In the **Attributes** section of the **General** tab, make sure **Read-only** is not checked.
6. Click **OK** in the Properties dialog.

Open the Image

1. Open **testdem.img** in the Viewer.

   This is a DEM file of the Gainesville, Georgia area, corresponding to the **lanier.img** data you have been using.

2. If it is not already displayed, select **AOI | Tools** from the Viewer menu bar to open the AOI tool palette.

   The AOI tool palette displays. The AOI tools are used to define the area(s) to be edited.

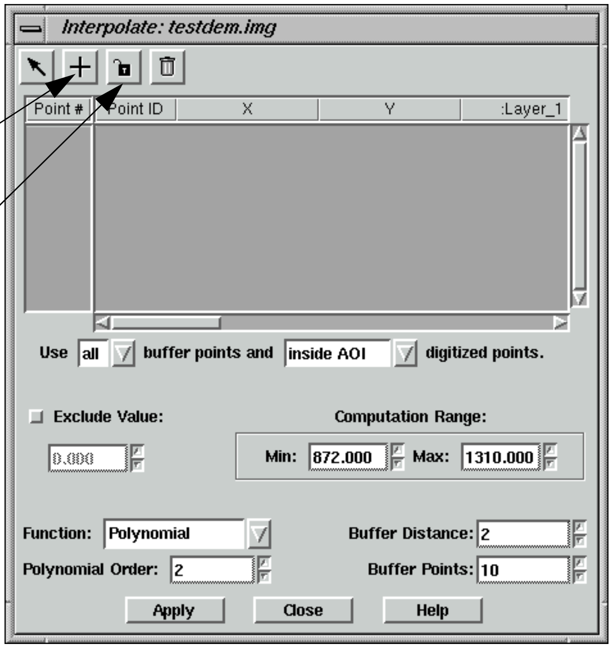
3. Click on the Ellipse icon in the AOI tool palette and then drag near the center of the Viewer image to draw an elliptical AOI, measuring about 1” to 2” in diameter.

When the mouse button is released, the AOI is surrounded by chaser lights and a bounding box.

Interpolate

1. In the Viewer menu bar, select **Raster | Interpolate**.

   The Interpolate dialog opens.



Use all buffer points and inside AOI digitized points.

☐ Exclude Value: 0.000

Computation Range: Min: 872.000 Max: 1310.000

Function: Polynomial

Polynomial Order: 2

Buffer Distance: 2

Buffer Points: 10

Click this icon to digitize points

Click this icon to lock the designated tool

Digitize Points

Digitizing points helps you to control the overall surface generated by the raster editing function. However, you should digitize points only when you know the general areas in the AOI that are bad. The Interpolation function uses the data values of the digitized points and the boundary of the AOI to generate a new surface.

1. Click on the Digitize Points icon in the Interpolate dialog, then click on the Lock icon.

2. Click at least 12 times in the AOI in the Viewer to digitize 12 points.

   The point coordinates display in the CellArray in the Interpolate dialog.

3. When you are finished digitizing points, click the Lock icon in the Interpolate dialog again to disable it.

4. In the Interpolate dialog under **Buffer Points**, enter **25** to allow up to 25 points in the computation.

5. In the Interpolate dialog under **Polynomial Order**, enter **3** to increase the polynomial order of interpolation.

6. Click **Apply** in the Interpolate dialog.

7. An Attention box displays, asking if you want to remove the data stretch lookup table. Click **Yes**.

8. A Warning box displays, suggesting that you recalculate the statistics. Click **OK**.

   The new surface displays inside the AOI.

9. Observe the changes in the AOI and then select **Raster | Undo** from the Viewer menu bar.

   The data values return to the original values. This lets you undo the edit without changing the original data values.

   NOTE: **Undo** *works only for the last edit applied.*

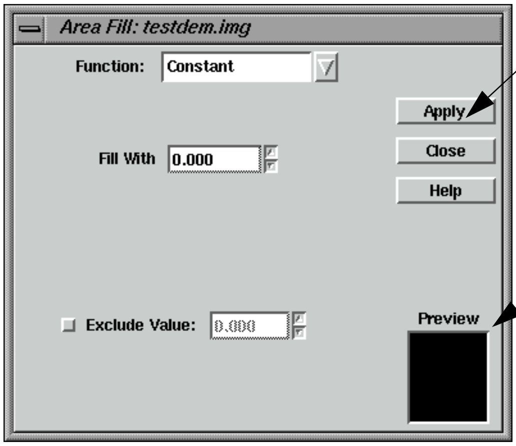
10. Click **Close** in the Interpolate dialog.

Fill with Constant Value

If the area to be edited is a flat surface, you may use a constant value to replace the bad data values.

1. Select **Raster | Fill** from the Viewer menu bar.

   The Area Fill dialog opens.



Area Fill: testdem.img

Function: Constant

Fill With 0.000

☐ Exclude Value: 0.000

Apply
Close
Help
Preview
Click here to apply the specified area fill
The preview window displays what the AOI fill looks like

2. In the Area Fill dialog, click **Apply** to accept the Constant function and its defaults.

   The AOI is replaced with a Constant value of zero—the area is black.

3. Select **Raster | Undo** from the Viewer menu bar.

   The image returns to the original values.

4. In the Area Fill dialog, enter **1500** in the **Fill With** number field and click **Apply**.

   Now the AOI fill area is white.

5. Select **Raster | Undo** from the Viewer menu bar.

   The image returns to the original values.

Set Global Value

1. In the Area Fill dialog, click on the **Function** popup list and select **Majority**.

   This option uses the majority of the pixel values in the AOI to replace all values in the AOI.

2. Click **Apply** in the Area Fill dialog.

   The AOI displays the newly generated surface.

3. After observing the changes, select **Raster | Undo** from the Viewer menu bar.
4. Click **Close** in the Area Fill dialog.
5. Select **File | Clear** from the Viewer menu bar.

   Save the AOI layer in the Viewer if you like.

Raster Attribute Editor

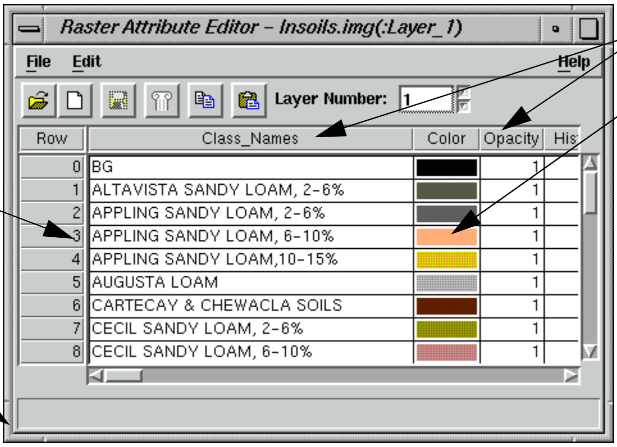
You can easily change the class colors in a thematic file. Here, you change the colors in **lnsoils.img**.

Change Color Attribute

Display **lnsoils.img** in a Viewer.

1. In the Viewer menu bar, select **Raster | Attributes**.

   The Raster Attribute Editor opens.



Layer Number: 1

Row	Class_Names	Color	Opacity	His
0	BG	⬛	1	
1	ALTAVISTA SANDY LOAM, 2-6%	⬛	1	
2	APPLING SANDY LOAM, 2-6%	⬛	1	
3	APPLING SANDY LOAM, 6-10%	🟧	1	
4	APPLING SANDY LOAM,10-15%	🟨	1	
5	AUGUSTA LOAM	⬜	1	
6	CARTECAY & CHEWACLA SOILS	🟫	1	
7	CECIL SANDY LOAM, 2-6%	🟩	1	
8	CECIL SANDY LOAM, 6-10%	🟥	1	

Column titles

Right-hold here for a list of colors

Select class to edit here or by clicking in the Viewer

Expand the CellArray by dragging on any corner

The CellArray in the Raster Attribute Editor is for manipulating the raster attributes and selecting classes to edit. To change the color of a class, you can select that class in two ways:

- with your cursor in the Viewer, click on the class you want to edit, or
- with your cursor in the Row column of the Raster Attribute Editor CellArray, click the class to edit.

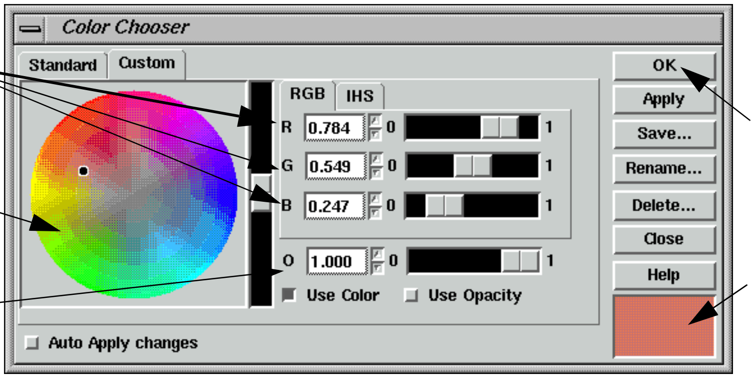
You use both methods in the following examples.

2. Move your cursor inside the Viewer and click on an area.

That class is highlighted in yellow in the Raster Attribute Editor CellArray, and the current color assigned to that class is shown in the bar underneath the **Color** column.

3. In the CellArray, right-hold with your cursor over the **Color** patch for the selected class and select **Other**.

The Color Chooser dialog opens.



R 0.784 0 1

G 0.549 0 1

B 0.247 0 1

O 1.000 0 1

☑ Use Color ☐ Use Opacity

☐ Auto Apply changes

Select colors here

Colorwheel

Enable transparent layers here

Click here to make the new color display in the Viewer

A preview of the selected color displays here

A dot is present on the colorwheel itself, indicating the current color of the selected class. This color is also shown in the preview window in the lower, right corner of the dialog.

4. In the Color Chooser dialog, change the color of the selected class by dragging the dot on the colorwheel to another spot on the colorwheel. Then, click the **Apply** button.

The selected class changes color in both the Viewer image and the Raster Attribute Editor CellArray.

You can also change the class color using any of these methods:

a) enter RGB (red, green, blue) or IHS (intensity, hue, saturation) values in the Color Selector number fields in the Color Chooser dialog, or

b) click the **Standard** *tab in the Color Chooser dialog to select from a list of predefined colors, or*

c) move the slider bars in the Color Chooser dialog

5. In the Raster Attribute Editor, select **Edit | Undo Last Edit**.

The change you made in step 4. is undone.

6. Click **Close** in the Color Chooser dialog.
7. Select **File | Close** from the Raster Attribute Editor.

Make Layers Transparent

If you have more than one file displayed in a Viewer, you can make specific classes or entire files transparent. In this example, you make the overlaid soils partially transparent so that the Landsat TM information shows through.

1. Display **lanier.img** over **lnsoils.img** in a Viewer. Be sure that the **Clear Display** checkbox is disabled under **Raster Options** when you are in the Select Layer To Add dialog.
2. In the Viewer menu bar, select **View | Arrange Layers**.

The Arrange Layers dialog opens.

3. In the Arrange Layers dialog, drag the **lnsoils.img** box on top of the **lanier.img** box.
4. Click **Apply**, then **Close** in the Arrange Layers dialog.

Edit Raster Attributes

1. Select **Raster | Attributes** from the Viewer menu bar.

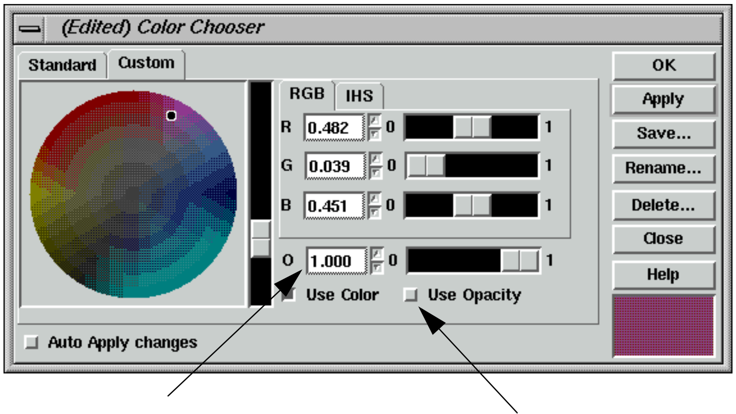
The Raster Attribute Editor displays.

The objective is to select a class that covers a section of **lanier.img** that you would like to see through **lnsoil.img**. Then, you can make that class transparent.

2. Select the class to become transparent, either by clicking in the Viewer or in the **Row** column of the CellArray.

3. In the Raster Attribute Editor CellArray, right-hold on the color button in the **Color** column of the selected class and drag to select **Other** from the popup list that displays.

   The Color Chooser dialog opens.



R 0.482 0 1
G 0.039 0 1
B 0.451 0 1
O 1.000 0 1

Click here and enter .40, or use the slider bar at right to set the number field to .40

Click checkbox to use opacity

4. In the Color Chooser dialog, click on the **Use Opacity** checkbox.

5. At **O** (which stands for Opacity), change the number to **.40** (opacity percentage of 40) using either the number field or the slider bar.

6. Click **Apply** in the Color Chooser dialog.

   The selected color becomes partially transparent, allowing you to see **lanier.img** underneath.

7. Experiment with different ways to change class color and opacity.

8. When you are finished, click **Close** in the Color Chooser dialog.

Manipulate CellArray Information

1. With your cursor in the title bar of the Raster Attribute Editor, drag it to the top of your screen.

2. Drag one of the bottom corners of the Raster Attribute Editor down until all rows of the CellArray are visible.

3. Drag the corners of the Raster Attribute Editor horizontally until all columns are visible.

   NOTE: The CellArray probably occupies most of your screen.

Select Rows

To select one row, simply click in the **Row** column of the desired row. That row is highlighted in yellow. You can select sequential rows by middle-clicking in additional rows. Shift-click in a selected row to deselect a row. You can also select rows using the **Row Selection** menu that opens when you right-hold in the **Row** column.

Select Columns

To select one column, click in the title box of the desired column. That column is highlighted in blue. You can select multiple columns by middle-clicking in the title bar of additional columns. Shift-click in a selected column to deselect it.

Choose Column Options

Many column options are available from the **Column Options** menu, which opens when you right-hold in a column title bar. You can have multiple columns and rows selected at the same time.

You use many of these features in the following steps.

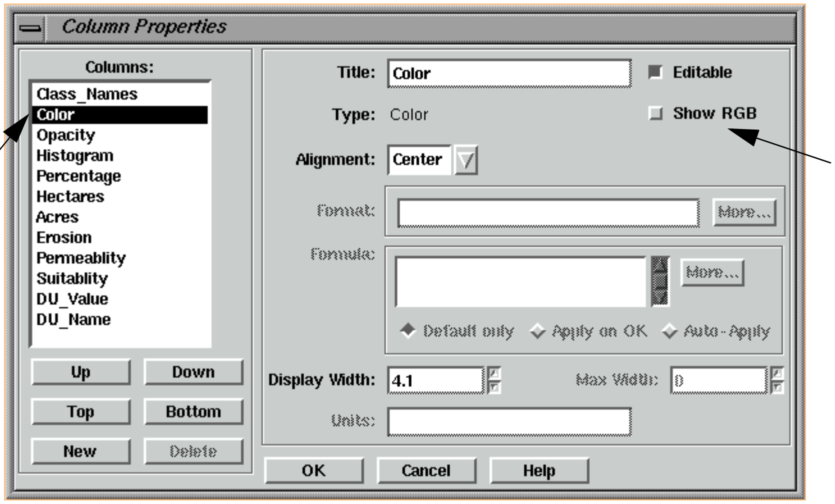
Resize Columns

You can make each column in the CellArray narrower and then reduce the width of the entire

Edit Column Properties

1. Click on the Column icon in the Raster Attribute Editor.

The Column Properties dialog opens.


Column Properties
Columns:
Class_Names
Color
Opacity
Histogram
Percentage
Hectares
Acres
Erosion
Permeablity
Suitablity
DU_Value
DU_Name
Up
Down
Top
Bottom
New
Delete
Editable
Show RGB
Format:
More...
Formula:
More...
Default only
Apply on OK
Auto-Apply
Units:
OK
Cancel
Help

Title: Color

Type: Color

Alignment: Center

Display Width: 4.1

Max Width: 0

Click here to select the Color column

Click here to activate this function

2. In the Column Properties dialog, select **Color** under **Columns** and activate the **Show RGB** checkbox.

3. Click **OK** in the Column Properties dialog.

4. In the Raster Attribute Editor, place your cursor on the column separator in the header row between the **Color** and **Red** columns.

   The cursor changes from the regular arrow to a double-headed arrow. You can now change the size of the **Color** column.

5. Drag the double-headed arrow to the right to make the **Color** column wider.

6. Repeat this procedure, dragging the double-headed arrow to the left, to narrow the other columns.

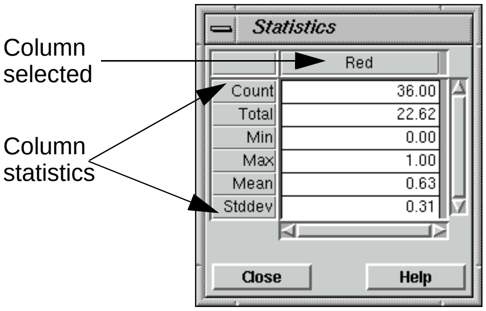
Generate Statistics

1. In the Raster Attribute Editor CellArray, select the entire **Red** column by clicking in the **Red** title box.

   The entire column is highlighted in blue.

2. With your cursor in the **Red** title box, right-hold **Column Options | Compute Stats**.

   The Statistics dialog opens.



	Red
Count	36.00
Total	22.62
Min	0.00
Max	1.00
Mean	0.63
Stddev	0.31

The Statistics for the column selected are reported.

Column Statistics

These statistics include:

- **Count**—number of classes selected
- **Total**—sum of column figures (in this example, total area)
- **Min**—minimum value represented in the column
- **Max**—maximum value represented in the column
- **Mean**—average value represented (**Total/Count**)
- **Stddev**—standard deviation

3. Click **Close** in the Statistics dialog.

Select Criteria

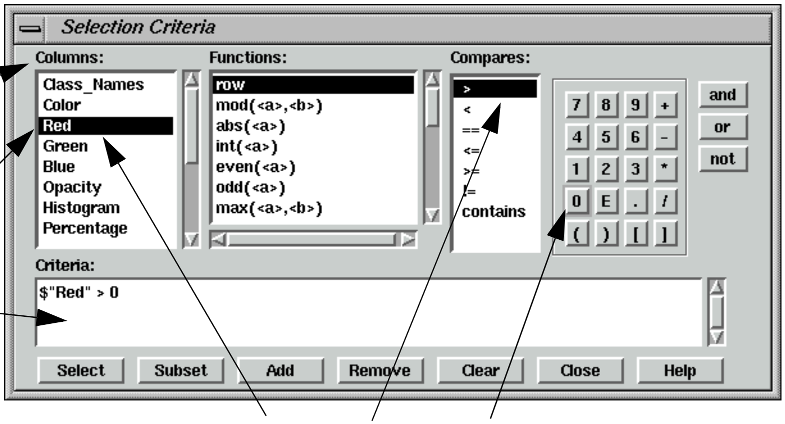
1. In the Raster Attribute Editor CellArray, select the **Class_Names** column by shift-clicking in the **Class_Names** title box.

   Now the **Class_Names** and **Red** columns are both selected; both columns are highlighted in blue.

   Next, you generate a report that lists all of the classes and the area covered by each. You do not include classes with an area of 0 (zero).

2. With your cursor in the **Row** column (not the header row of the **Row** column), right-hold **Row Selection | Criteria**.

   The Selection Criteria dialog opens.


Selection Criteria
Columns:
Class_Names
Color
Red
Green
Blue
Opacity
Histogram
Percentage
Functions:
row
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
max(<a>,<b>)
Compares:
>
<
==
<=
>=
!=
contains
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
and
or
not
Criteria:
Select
Subset
Add
Remove
Clear
Close
Help

$"Red" > 0

All columns in the CellArray are listed here

Click here and it displays here

Click here, then here, then here to define the criteria (automatically written to the Criteria definition box)

3. In the Selection Criteria dialog under **Columns**, click **Red**.

   $“Red” is written in the **Criteria** definition box at the bottom of the dialog.

4. Under **Compares**, click **>**.

5. In the calculator, click **0**.

   The Criteria should now read:

   $ “Red” > 0

   This criteria selects all classes in the CellArray with an area that is greater than 0.

6. Click **Select** in the Selection Criteria dialog to select these rows in the CellArray.

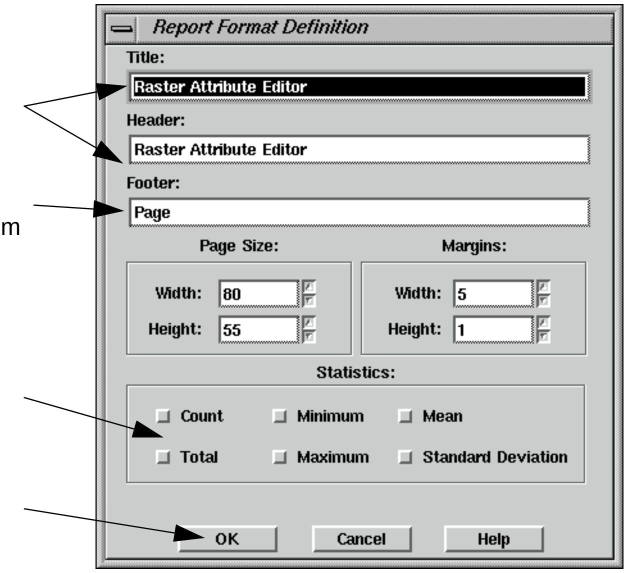
   All rows except 0, 12, and 34 are selected (i.e., highlighted in yellow). These rows are not selected because the opacity for each of these categories is 0.

7. Click **Close** in the Selection Criteria dialog.

Generate Report

1. With your cursor in a **Class_Names** title box, right-hold **Column Options | Report**.

   The Report Format Definition dialog opens.



Report Format Definition

Title: Raster Attribute Editor

Header: Raster Attribute Editor

Footer: Page

Page Size: Width: 80 Height: 55

Margins: Width: 5 Height: 1

Statistics:

☐ Count ☐ Minimum ☐ Mean

☐ Total ☐ Maximum ☐ Standard Deviation

Enter report title and header here

Enter text to precede the page number on the bottom of the report

Specify statistics to include here

Click here to generate a report

2. In the Report Format Definition dialog under **Title**, add **for lnsoils.img** to the default text string.

3. Under **Header**, add **for lnsoils.img** to the default text string.

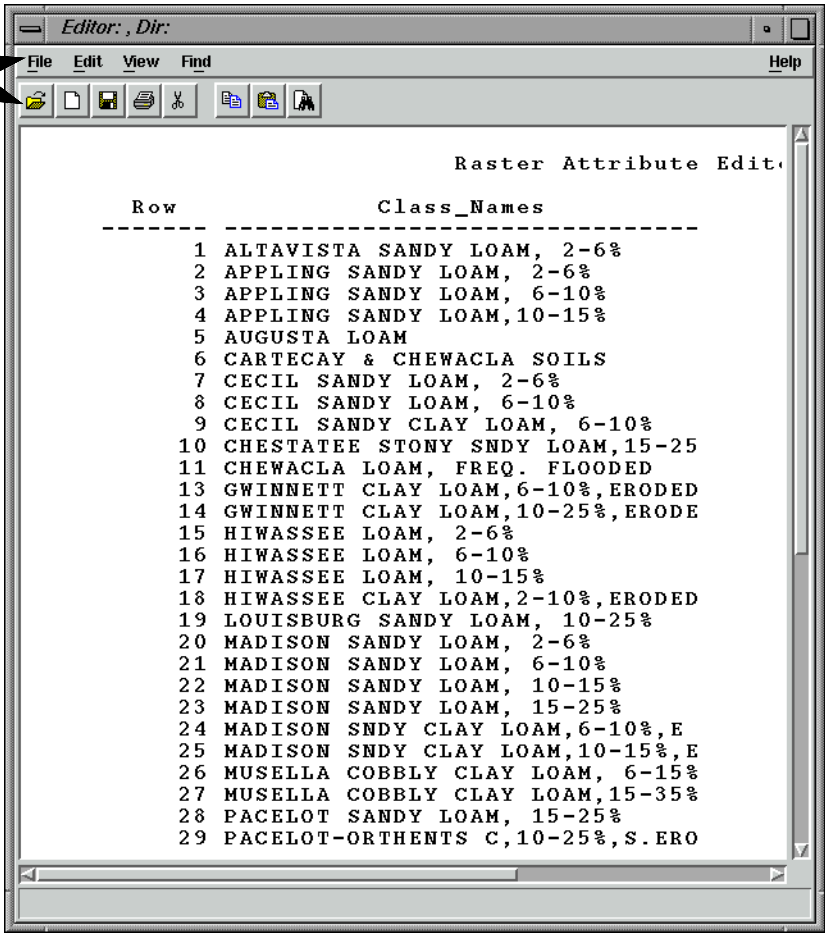
4. Under **Statistics**, click on each checkbox to include all available statistics in the report.

5. Click **OK** in the Report Format Definition dialog to generate the report.

   A Job Status dialog opens, indicating the progress of the function.

   When the function is complete, the report displays in an IMAGINE Text Editor.

Use these menu bar and tool bar items to save and edit report



```
                                 Raster Attribute Edit
    Row            Class_Names
  ------- ------------------------------
        1 ALTAVISTA SANDY LOAM, 2-6%
        2 APPLING SANDY LOAM, 2-6%
        3 APPLING SANDY LOAM, 6-10%
        4 APPLING SANDY LOAM,10-15%
        5 AUGUSTA LOAM
        6 CARTECAY & CHEWACLA SOILS
        7 CECIL SANDY LOAM, 2-6%
        8 CECIL SANDY LOAM, 6-10%
        9 CECIL SANDY CLAY LOAM, 6-10%
       10 CHESTATEE STONY SNDY LOAM,15-25
       11 CHEWACLA LOAM, FREQ. FLOODED
       13 GWINNETT CLAY LOAM,6-10%,ERODED
       14 GWINNETT CLAY LOAM,10-25%,ERODE
       15 HIWASSEE LOAM, 2-6%
       16 HIWASSEE LOAM, 6-10%
       17 HIWASSEE LOAM, 10-15%
       18 HIWASSEE CLAY LOAM,2-10%,ERODED
       19 LOUISBURG SANDY LOAM, 10-25%
       20 MADISON SANDY LOAM, 2-6%
       21 MADISON SANDY LOAM, 6-10%
       22 MADISON SANDY LOAM, 10-15%
       23 MADISON SANDY LOAM, 15-25%
       24 MADISON SNDY CLAY LOAM,6-10%,E
       25 MADISON SNDY CLAY LOAM,10-15%,E
       26 MUSELLA COBBLY CLAY LOAM, 6-15%
       27 MUSELLA COBBLY CLAY LOAM,15-35%
       28 PACELOT SANDY LOAM, 15-25%
       29 PACELOT-ORTHENTS C,10-25%,S.ERO
```

6. To save the report, in the Text Editor menu bar, select **File | Save As**.

   The Save As dialog opens.

7. Enter a name for the report, such as **soilsreport.txt**.

8. Click **OK** in the Save As dialog.

   The file name is written in the title bar of the Text Editor window.

9. In the Text Editor menu bar, select **File | Close**.

10. Select **File | Close** from the Raster Attribute Editor.

    Save your changes to the Raster Attribute Editor if you like.

11. In the Viewer tool bar, click the Erase icon to clear the window.

Profile Tools

The spectral profile display is fundamental to the analysis of hyperspectral data sets. As the number of bands increases and the band widths decrease, the remote sensor is evolving toward the visible/infrared spectrometer. The reflectance (DN) of each band within one (spatial) pixel can be plotted to provide a curve approximating the profile generated by a laboratory scanning spectrometer. This allows estimates of the chemical composition of the material in the pixel. To use this tool, follow the steps below.

Prepare

ERDAS IMAGINE should be running and a Viewer should be open.

Display Spectral Profile

1. In the Viewer menu bar, select **File | Open | Raster Layer**.

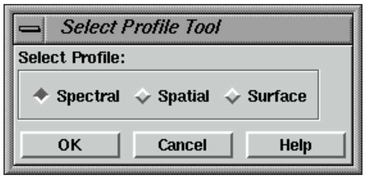
   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select **hyperspectral.img** under **Filename**.
3. Click the **Raster Options** tab at the top of the dialog.
4. In the **Raster Options**, click the **Fit to Frame** checkbox to activate it and then click **OK**.

   The file **hyperspectral.img** is displayed in the Viewer.

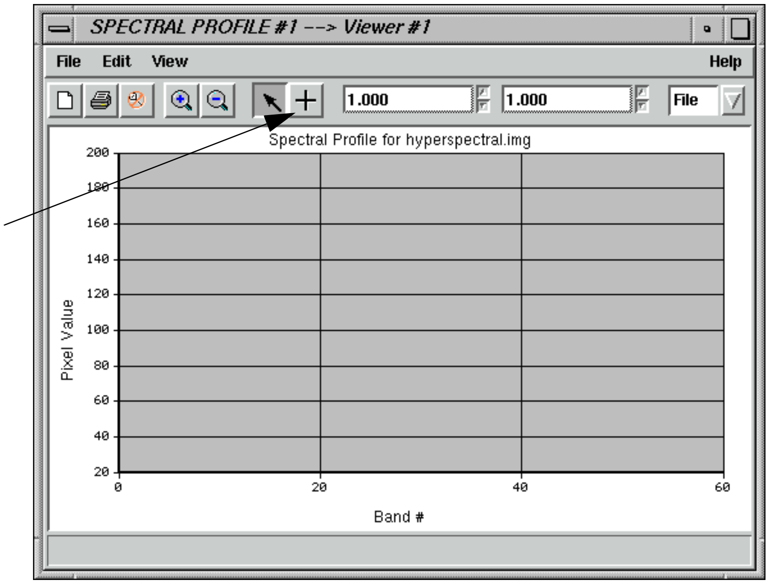
5. In the Viewer menu bar, select **Raster | Profile Tools**.

   The Select Profile Tool dialog opens.



6. Accept the **Spectral** default and click **OK** in the Select Profile Tool dialog.

   The Spectral Profile viewer opens.


SPECTRAL PROFILE #1 --> Viewer #1
File
Edit
View
Help
1.000
1.000
File
Spectral Profile for hyperspectral.img
Pixel Value
200
180
160
140
120
100
80
60
40
20
0
20
40
60
Band #

Click here to create a new profile point in the Viewer

7. In the Spectral Profile viewer, click on the Create icon and then select a pixel of interest by clicking it in the Viewer image.

The data for the selected pixel are displayed in the Spectral Profile viewer.

NOTE: The pixel can be moved around the displayed image by dragging it.

Analyze Data

1. In the Spectral Profile viewer menu bar, select **Edit | Chart Options**.

The Chart Options dialog opens.

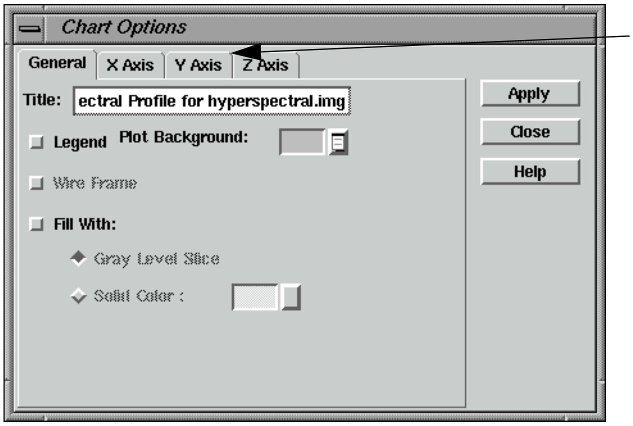


Chart Options
General
X Axis
Y Axis
Z Axis
Legend
Plot Background:
Wire Frame
Fill With:
Gray Level Slice
Solid Color :
Apply
Close
Help

Title: ectral Profile for hyperspectral.img

Click this tab to view the options for the Y axis

2. In the Chart Options dialog, click on the **Y Axis** tab at the top of the dialog.

3. Set **Min** to **20** and **Max** to **180** to control the numerical range.

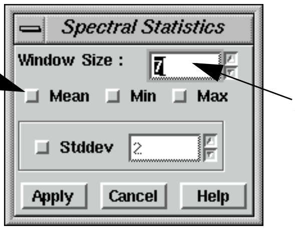
4. Click **Apply** and then **Close** in the Chart Options dialog.

   The selected range is shown in detail in the Spectral Profile viewer.

5. In the Spectral Profile viewer menu bar, select **Edit | Plot Stats**.

   The Spectral Statistics dialog opens.

Click to select Mean


Spectral Statistics
Mean
Min
Max
Stddev
Apply
Cancel
Help

Window Size : 7

Stddev: 2

Adjust the window size here

6. In the Spectral Statistics dialog, change the **Window Size** to **7**.

7. Select **Mean** and click **Apply**.

   The mean (within the selected window) of Profile 1 is depicted on the graph.

8. Click **Cancel** in the Spectral Statistics dialog.

9. Select **File | Close** from the Spectral Profile viewer.

Wavelength Axis

Data tapes containing hyperspectral imagery commonly designate the bands as a simple numerical sequence. When plotted using the profile tools, this yields an x-axis labeled as 1, 2, 3, 4, etc. Elsewhere on the tape or in the accompanying documentation is a file which lists the center frequency and width of each band. This information should be linked to the image intensity values for accurate analysis or comparison to other spectra, such as the Spectra Libraries.

To do this, the band position information must be entered into a linkable format, which is an **.saf** file. An example of this format can be seen by using a **texteditor** or **vi** command to view one of the **.saf** files in <IMAGINE_HOME>/etc. Once this file is created, it can be linked with the Spectral Profile by using the **Edit | Use Sensor Attributes** option.

Display Spatial Profile

The Spatial Profile display function allows the analyst to view the reflectance(s) of the pixels along a user-defined polyline. The display can be viewed in either two-dimensional (one band) or perspective three-dimensional (multiple bands) mode. To use this tool, follow the steps below.

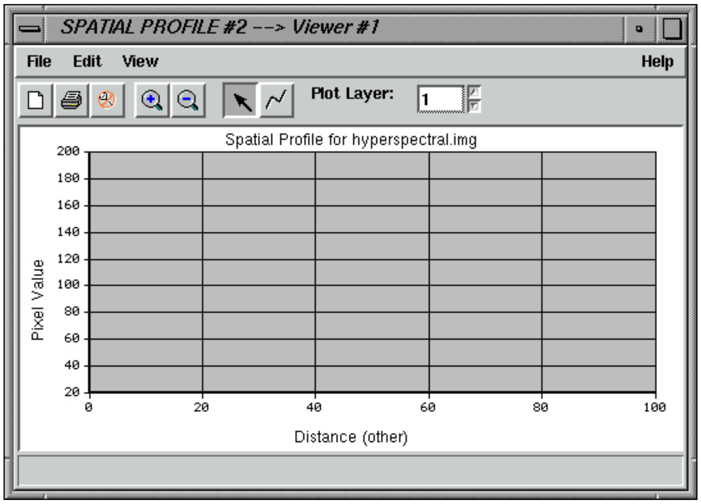
The file **hyperspectral.img** should be displayed in a Viewer, with the **Fit to Frame** checkbox activated.

1. In the Viewer menu bar, select **Raster | Profile Tools**.

   The Select Profile Tool dialog opens.

2. Click on the **Spatial** button in the Select Profile Tool dialog and then click **OK**.

   The Spatial Profile viewer opens.


SPATIAL PROFILE #2 --> Viewer #1
File
Edit
View
Help
Plot Layer: 1
Spatial Profile for hyperspectral.img
Pixel Value
200
180
160
140
120
100
80
60
40
20
0
20
40
60
80
100
Distance (other)

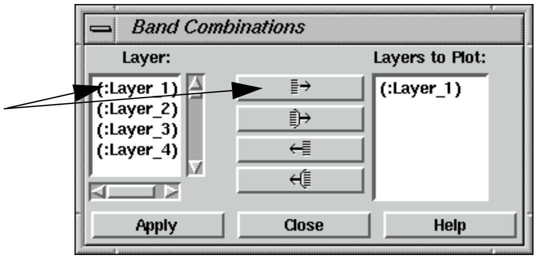
3. Click on the Polyline icon in the Spatial Profile viewer tool bar and then draw a polyline on the image in the Viewer. Click to set vertices and middle-click to set an endpoint.

The spatial profile is displayed in the Spatial Profile viewer.

Analyze Data

1. Select **Edit | Plot Layers** from the Spatial Profile viewer menu bar.

The Band Combinations dialog opens.



Click here and
then here
to add the layer
to the plot

2. Add Layers **2** and **3** to the **Layers to Plot** column by individually selecting them under **Layer** and clicking the Add Selected Layer button.

3. Click **Apply** and then **Close** in the Band Combinations dialog.

Layers 1, 2, and 3 are plotted in the Spatial Profile viewer.

NOTE: Moving the cursor around in the Spatial Profile viewer gives you the pixel values for the x and y coordinates of the layers.

4. In the **Plot Layer** box to the right of the tool bar in the Spatial Profile viewer, click on the up arrow to view layers **4** and **5**.

5. Select **Edit | Plot Layers** from the Spatial Profile viewer to again bring up the Band Combinations dialog.

6. In the Band Combinations dialog, click the Add All button.

7. Click **Apply** and **Close**.

   As in the Spectral Profile viewer, you can select **Edit | Chart Options** to optimize the display.

8. Select **File | Close** from the Spatial Profile viewer menu bar.

View Surface Profile

The Surface Profile can be used to view any layer (band) or subset in the data cube as a relief surface. To use this tool, follow the steps below.

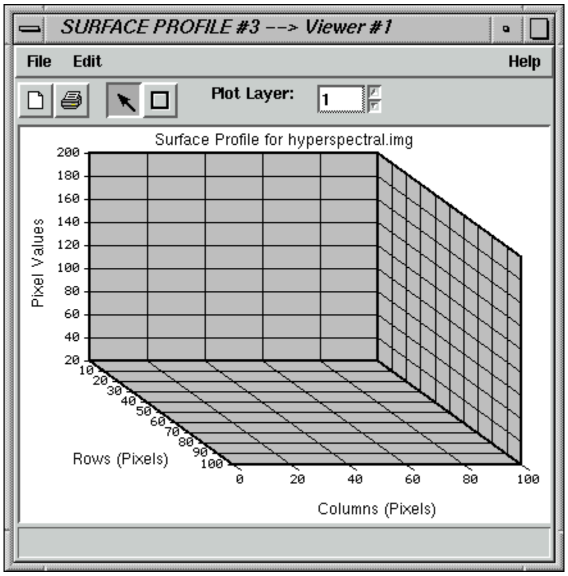
The file **hyperspectral.img** should be displayed in a Viewer with the **Fit to Frame** checkbox activated.

1. In the Viewer menu bar, select **Raster | Profile Tools**.

   The Select Profile Tool dialog opens.

2. In the Select Profile Tool dialog, click the **Surface** button and then click **OK**.

   The Surface Profile viewer opens.


SURFACE PROFILE #3 --> Viewer #1
File
Edit
Help
Surface Profile for hyperspectral.img
Pixel Values
200
180
160
140
120
100
80
60
40
20
10
20
30
40
50
60
70
80
90
100
Rows (Pixels)
0
20
40
60
80
100
Columns (Pixels)

Plot Layer: 1

3. Click on the Rectangle icon in the Surface Profile viewer and then select an AOI in the Viewer by dragging to create a box around it.

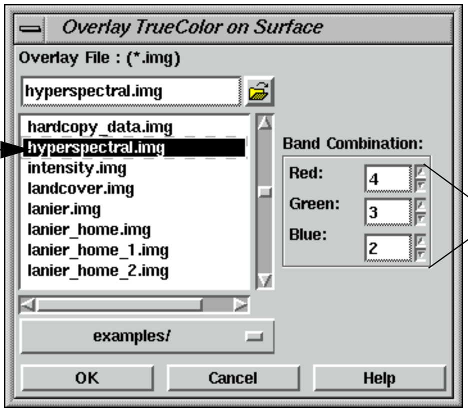
   When the mouse button is released, the surface profile for the selected area is displayed in the Surface Profile viewer. As with all of the profile tools, selecting **Edit | Chart Options** allows you to optimize the display.

Analyze Data

It may be desirable to overlay a thematic layer onto this surface. For example, a vegetation map could be overlaid onto a DEM surface, or an iron oxide map (Landsat TM3/TM1) onto a kaolinite peak (1.40 µm) layer. In this example, you overlay a true color image.

1. In the Surface Profile viewer, select **Edit | Overlay True Color**.

   The Overlay TrueColor on Surface dialog opens.


Overlay TrueColor on Surface
hardcopy_data.img
hyperspectral.img
intensity.img
landcover.img
lanier.img
lanier_home.img
lanier_home_1.img
lanier_home_2.img
examples/
OK
Cancel
Help

Overlay File : (*.img) hyperspectral.img

Band Combination:

Red: 4

Green: 3

Blue: 2

Click to select the file

Change the RGB band combinations here

2. In the Overlay TrueColor on Surface dialog, select **hyperspectral.img** under **Overlay File**.
3. Under **Band Combination**, enter **55** for **Red**, **34** for **Green**, and **2** for **Blue**.
4. Click **OK** in the Overlay TrueColor on Surface dialog.
5. When you are finished analyzing the data, select **File | Close** from the Spatial Profile viewer menu bar.

For more information on Hyperspectral Image Processing or the Hyperspectral Profile Tools, see "Chapter 5: Enhancement" *in the* ERDAS Field Guide.

Image Drape

It is possible to access the Image Drape utility either through the **Tools** menu in the ERDAS IMAGINE menu bar or through the Viewer. Here, you access the Image Drape utility via the Viewer.

ERDAS IMAGINE should be running and a Viewer should be open.

1. Click the Open icon in the Viewer tool bar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, select the file **eldodem.img**.

3. Click **OK** in the Select Layer To Add dialog.

   The file **eldodem.img** displays in the Viewer.

4. Click the Open icon again in the Viewer tool bar.

   The Select Layer To Add dialog reopens.

5. In the Select Layer To Add dialog under **Filename**, select the file **eldoatm.img**.

6. Click the **Raster Options** tab at the top of the dialog.

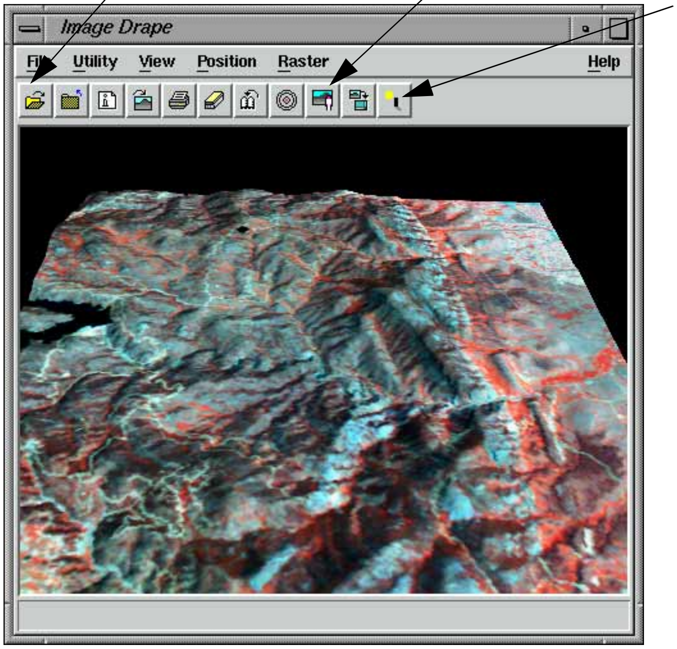
7. In the Raster Options, click the **Clear Display** checkbox to turn it off. This allows **eldoatm.img** to display on top of **eldodem.img**.

8. Click **OK** in the Select Layer To Add dialog.

   Now, both **eldodem.img** and **eldoatm.img** are displayed in the same Viewer, with the **eldoatm.img** layer on top.

9. Select **Utility | Image Drape** from the Viewer menu bar.

   An Image Drape viewer displays, with the overlapping images in it.



Click this icon to open a file in the Image Drape viewer

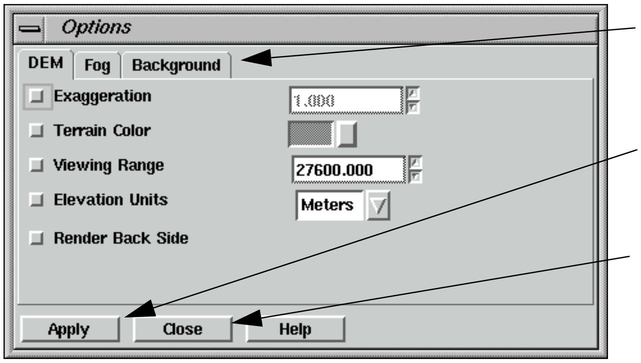
Click on this icon to open the Observer Positioning tool

Click on this icon to open the Sun Positioning tool

Change Options

1. Select **Utility | Options** from the Image Drape viewer menu bar.

   The Options dialog opens.


Options
DEM
Fog
Background
Exaggeration
1.000
Terrain Color
Viewing Range
27600.000
Elevation Units
Meters
Render Back Side
Apply
Close
Help

Click on this tab to edit background options

Click to apply your changes and close this dialog

Click to apply no changes and close this dialog

2. Click on the **Background** tab in the Options dialog.
3. In the Background options, hold on the popup list next to **Background Color** and select **Gold**.
4. Click **Apply** in the Options dialog.

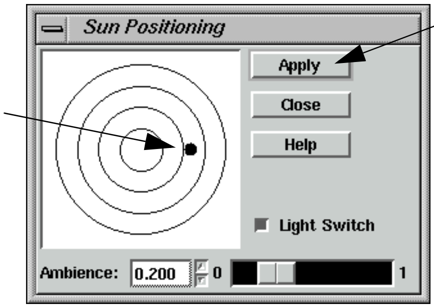
   The background of the image in the Image Drape viewer is now gold.

5. Click **Close** in the Options dialog.

Change Sun Position

1. Select **View | Sun Positioning** from the Image Drape viewer menu bar.

   The Sun Positioning dialog opens.


Sun Positioning
Apply
Close
Help
Light Switch
Ambience:
0.200
0
1

Click here to apply your changes

Drag this dot to change the position of the sun

2. In the Sun Positioning dialog, drag the dot to another position on the target. The center of the target indicates the sun position at high noon.

3. Click **Apply** and then **Close** in the Sun Positioning dialog.

   Note how the shadows across the image change to reflect the different sun position you have selected.

Dump Contents to Viewer

1. Select **Utility | Dump Contents to Viewer** from the Image Drape viewer menu bar.

   A second Viewer opens, displaying another view of the image in the Image Drape viewer.

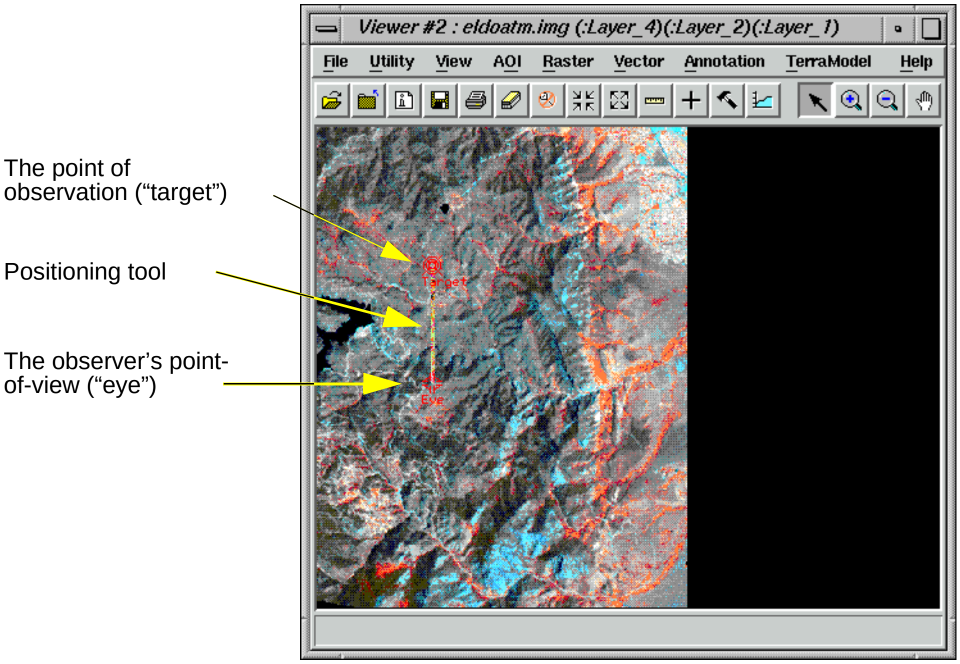
2. Select **File | Close** in the first Viewer to clear it from the screen.

3. Select **View | Link/Unlink with Viewer** from the Image Drape viewer menu bar.

   An instructions box opens, directing you to click in the Viewer to which you want the Image Drape viewer to be linked.

4. Click in the Viewer you just created.

   The viewers are now linked and a Positioning tool displays in the Viewer.

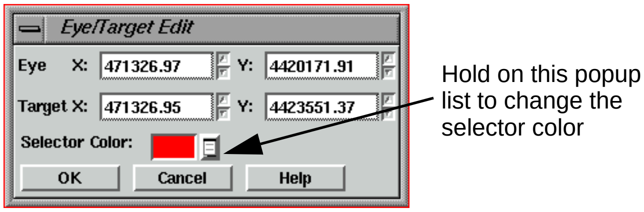

Viewer #2 : eldoatm.img (:Layer_4)(:Layer_2)(:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
Target
Eye
The point of observation (“target”)
Positioning tool
The observer’s point-of-view (“eye”)

NOTE: The bounding box in the Viewer image pictured above is for visual purposes only, and does not actually appear in the Viewer window.

Start Eye/Target

1. To make the Positioning tool easier to see in the Viewer, select **Utility | Selector Properties** from the Viewer menu bar.

The Eye/Target Edit dialog opens.



Eye/Target Edit

Eye X: 471326.97 Y: 4420171.91

Target X: 471326.95 Y: 4423551.37

Selector Color:

Hold on this popup list to change the selector color

2. In the Eye/Target Edit dialog, hold on the **Selector Color** popup list and select a color that displays well in the Viewer image (e.g., Yellow).

3. Click **OK** in the Eye/Target Edit dialog.

   The dialog closes and the color of the Positioning tool is updated to the designated color.

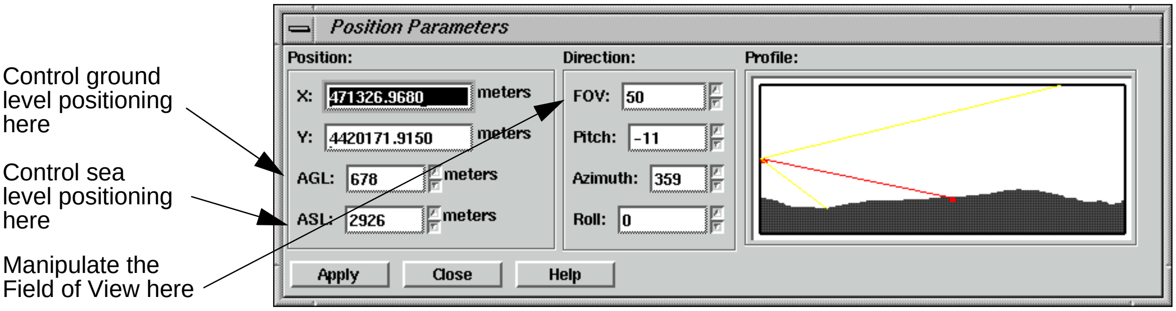
Manipulate the Observer and Field of View

You can manipulate the observer and the observer's field of view in several ways. See steps (1. through step 3. below to learn how to obtain different views using the Positioning tool and the Position Parameters dialog.

Use the Positioning Tool

1. Click on the Observer Positioning icon in the Image Drape viewer tool bar.

   The Position Parameters dialog opens.



Position Parameters

Position:

X: 471326.9680 meters

Y: 4420171.9150 meters

AGL: 678 meters

ASL: 2926 meters

Direction:

FOV: 50

Pitch: -11

Azimuth: 359

Roll: 0

Control ground level positioning here

Control sea level positioning here

Manipulate the Field of View here

2. Drag on the **Eye** marker of the Positioning tool to change the observer's point of view in the Image Drape viewer.

   The data in the Position Parameters dialog updates to reflect the changes in the observer's position. The view in the Image Drape viewer is also updated.

If the image in the Image Drape viewer does not completely refresh when the mouse button is released, click the Update icon.

3. Next, drag on the **Target** marker of the Positioning tool to change the point of observation in the Image Drape viewer.

   The data in the Position Parameters dialog updates to reflect the changes in the point of observation. The view in the Image Drape viewer is also updated.

4. Change the position of both the observer and the target at once by dragging on the line that connects them in the Viewer.

   The data in the Position Parameters dialog and the Image Drape viewer is updated.

5. In the Image Drape viewer, click the Goto icon to return to the original position of the observer and the target.

   The image and the Positioning tool in the Viewer are updated to their original position. The Position Parameters dialog is also updated.

Use the Position Parameters Dialog

1. In the Position Parameters dialog, change the **FOV** to **90** and the **Roll** to **45**, then click **Apply**.

   The image in the Image Drape viewer is updated to reflect this change.

2. In the Position Parameters dialog under **Observer Position**, enter **3000** in the **AGL** (Above Ground Level) number field.

3. Click **Apply** and then **Close** in the Position Parameters dialog.

CHAPTER TWO

Image Catalog

Introduction

In this tour guide, you can learn how to:

- set catalog preferences
- create an Image Catalog
- add information to a catalog
- perform queries
- view the information in a catalog
- modify the views/information
- save and restore a specific area of coverage
- archive data
- add custom maps

Approximate completion time for this tour guide is 15 minutes.

Set Catalog Preferences

ERDAS IMAGINE should be running. It is not necessary to have a Viewer open.

1. In the ERDAS IMAGINE menu bar, select **Session | Preferences**.

   The Preference Editor opens.

2. In the Preference Editor under the **Category** list, select **Image Catalog**.

   The default preferences for the Image Catalog display.

   The following table lists and describes the various catalog preferences.

Table 2-1: Image Catalog Preferences

Preference	**Function**
Catalog Directory	The default directory that the catalog uses to save and open catalogs.
Default Catalog	The name of the catalog that is opened by default when the Image Catalog starts.

Table 2-1: Image Catalog Preferences (Continued)

Canvas Backdrop	The backdrop file (ArcInfo coverage) that is displayed when the Graphical Query viewer is initially displayed. This must be one of the files named in catalog.cov.
Archive Media	The name of the default media used by the archive function of the catalog.
Water Color	The default color used for all of the area outside of the polygons in a coverage, which is assumed to be water.
Land Color	The default color used to fill the polygons in an area coverage.
Border Color	The default color used to draw the border around each of the polygons in an area coverage.
Line Style	The default style used for lines in a line coverage.
Point Symbol	The default symbol used for points in a point coverage.
Symbol Color	The default color used for point symbols in a point coverage.
Symbol Size	The default size of point symbols (in points) in a point coverage.
Footprint Color	The default color used for image footprints.
Footprint Selected Color	The default color used for selected image footprints.
Footprint Fill Style	The default style used to fill image footprints.
Show Map Grid	Used to control the display of the map grid at startup.
Grid Color	The default color used for the Lat/Lon grid.
Show Grid Labels	Used to control the display of the text grid labels.
Grid Label Color	The default color used for the map grid text labels.
Level of Detail	Maximum level of detail to use when displaying maps (e.g., local, global).
Show Map Outlines	Enable to display the map outlines at startup.
Map Outline Color	The default color used to display the map outlines.
Map Outline Style	The default line style used for the map outlines (e.g., solid, dashed).
Restore Directory	The default destination directory for restoring an image.

3. Make any changes you wish to the catalog preferences by entering the new information in the fields and pressing Return on your keyboard.

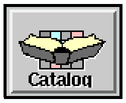
4. To save your changes and accept them as the new defaults, click **User Save**.

5. Click **Close** on the Preference Editor dialog to close the Preference Editor.

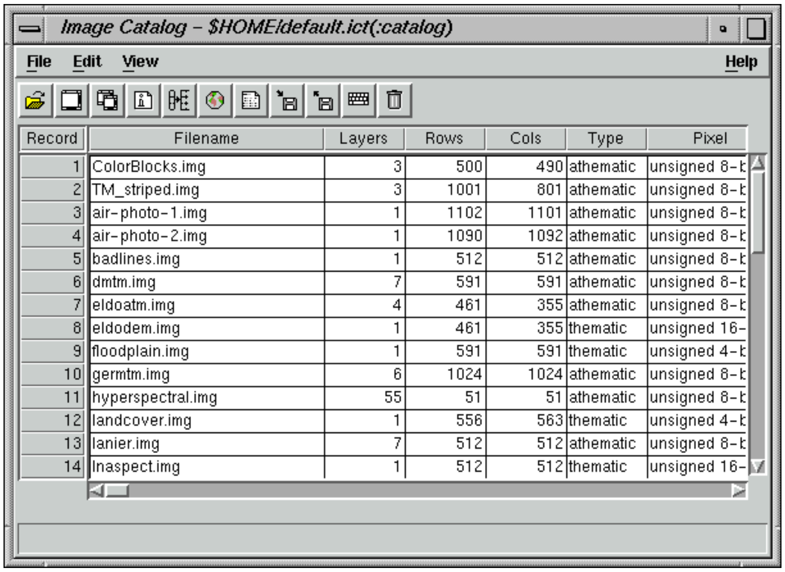
Create an Image Catalog

ERDAS IMAGINE should be running.

1. Click the Catalog icon in the ERDAS IMAGINE icon panel.



The default Image Catalog opens.



Record	Filename	Layers	Rows	Cols	Type	Pixel
1	ColorBlocks.img	3	500	490	athematic	unsigned 8-
2	TM_striped.img	3	1001	801	athematic	unsigned 8-
3	air-photo-1.img	1	1102	1101	athematic	unsigned 8-
4	air-photo-2.img	1	1090	1092	athematic	unsigned 8-
5	badlines.img	1	512	512	athematic	unsigned 8-
6	dmtm.img	7	591	591	athematic	unsigned 8-
7	eldoatm.img	4	461	355	athematic	unsigned 8-
8	eldodem.img	1	461	355	thematic	unsigned 16-
9	floodplain.img	1	591	591	thematic	unsigned 4-
10	germtm.img	6	1024	1024	athematic	unsigned 8-
11	hyperspectral.img	55	51	51	athematic	unsigned 8-
12	landcover.img	1	556	563	thematic	unsigned 4-
13	lanier.img	7	512	512	athematic	unsigned 8-
14	lnaspect.img	1	512	512	thematic	unsigned 16-

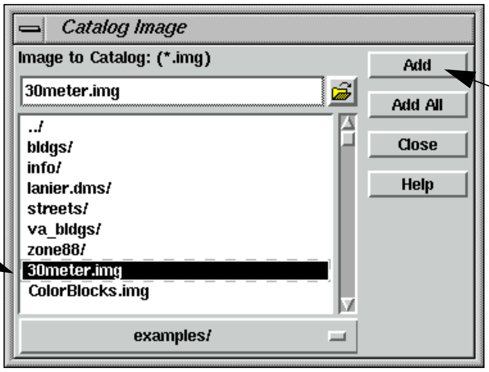
NOTE: You can specify a default catalog via **Session | Preferences** *in the ERDAS IMAGINE menu. More information on how to do this is located in* "Set Catalog Preferences" *on page 59.*

For the following example, however, use the default catalog which is distributed with ERDAS IMAGINE.

Add Information

1. Select **Edit | Catalog Image** or click on the Add Image icon in the Image Catalog tool bar.

The Catalog Image dialog opens.


Catalog Image
Image to Catalog: (*.img)
30meter.img
../
bldgs/
info/
lanier.dms/
streets/
va_bldgs/
zone88/
30meter.img
ColorBlocks.img
examples/
Add
Add All
Close
Help

Click here to select this file to add to the Image Catalog

Click here to add the selected file to the Image Catalog

2. In the Catalog Image dialog file list, select **30meter.img**.
3. Click **Add**, and then **Close** in the Catalog Image dialog.

   30meter.img is recorded in the Image Catalog.

   NOTE: If you were to select **30meter.img** *and then select* **Add All**, *all files with an .img extension would be added to the Image Catalog.*

Perform Graphical Queries

First, select an area of the map in which you want to query the images, and then you run the query.

The Image Catalog should be open and it should contain the images in the <IMAGINE_HOME>/examples directory.

Select Area

1. Select **View | Graphical Query Viewer** or click the Visual Query icon in the Image Catalog tool bar.

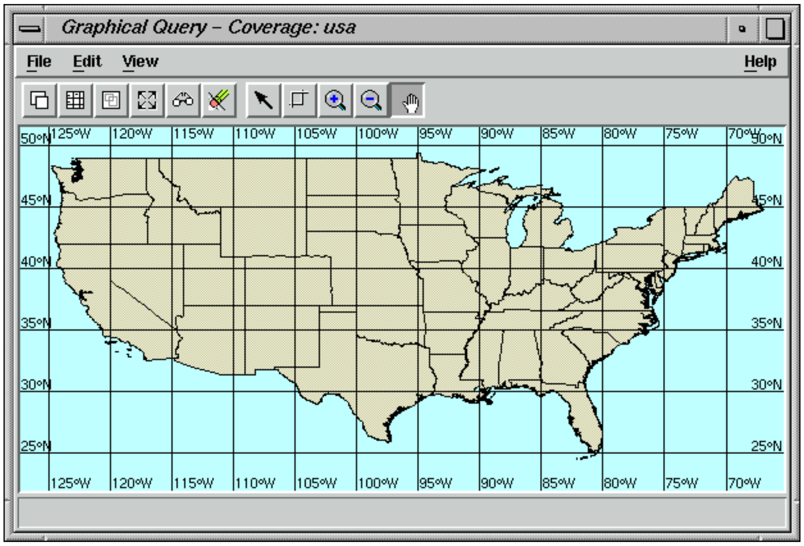
   The Graphical Query viewer opens.

2. Click the Select Area for Zooming icon in the Graphical Query viewer tool bar.

   The pointer becomes a crosshair.

3. With the crosshair, draw a bounding box that covers the entire continental United States.

   When the mouse button is released, the United States is magnified in the Graphical Query viewer. Note that more detail is presented as the backdrop is zoomed.


Graphical Query – Coverage: usa
File
Edit
View
Help
125°W
120°W
115°W
110°W
105°W
100°W
95°W
90°W
85°W
80°W
75°W
70°W
50°N
45°N
40°N
35°N
30°N
25°N

Run Query

1. Click the Query icon in the Graphical Query viewer tool bar.

   The files of images located within the designated query area (the continental US) are highlighted in yellow in the Image Catalog. Tiny rectangles on the map in the Graphical Query viewer mark the location of these images.

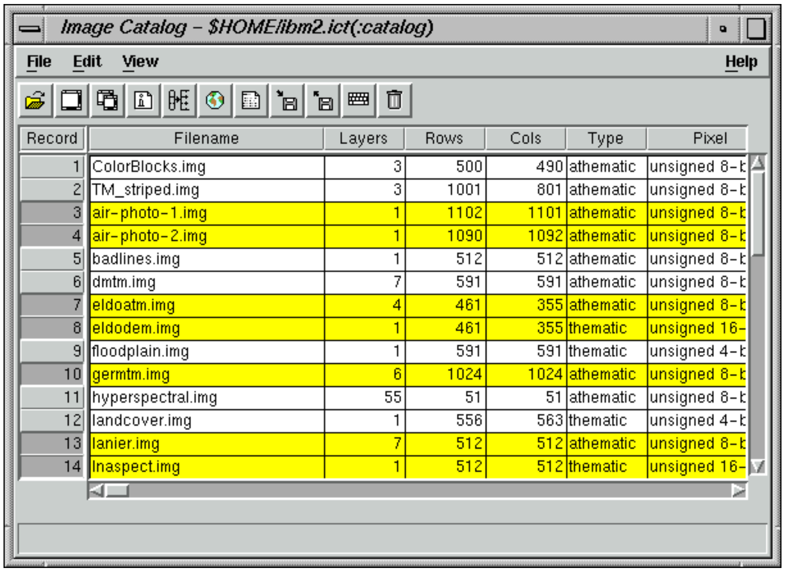


Image Catalog – $HOME/ibm2.ict(:catalog)
File
Edit
View
Help

Record	Filename	Layers	Rows	Cols	Type	Pixel
1	ColorBlocks.img	3	500	490	athematic	unsigned 8-
2	TM_striped.img	3	1001	801	athematic	unsigned 8-
3	air-photo-1.img	1	1102	1101	athematic	unsigned 8-
4	air-photo-2.img	1	1090	1092	athematic	unsigned 8-
5	badlines.img	1	512	512	athematic	unsigned 8-
6	dmtm.img	7	591	591	athematic	unsigned 8-
7	eldoatm.img	4	461	355	athematic	unsigned 8-
8	eldodem.img	1	461	355	thematic	unsigned 16-
9	floodplain.img	1	591	591	thematic	unsigned 4-
10	germtm.img	6	1024	1024	athematic	unsigned 8-
11	hyperspectral.img	55	51	51	athematic	unsigned 8-
12	landcover.img	1	556	563	thematic	unsigned 4-
13	lanier.img	7	512	512	athematic	unsigned 8-
14	lnaspect.img	1	512	512	thematic	unsigned 16-

2. Click the Zoom In icon in the Graphical Query viewer tool bar.
3. Place the Zoom In tool over one of the black dots in the map in the Graphical Query viewer and click until the dot is magnified to a yellow box.
4. Click the Select icon in the Graphical Query viewer tool bar.
5. With the Select tool, click on the yellow box you just magnified. Note that all of the other images located in the Graphical Query viewer are characterized by red boxes.

   The corresponding filename is highlighted in the Image Catalog.

6. Click the Display Single icon in the Image Catalog tool bar.

   The selected image displays in a Viewer.

7. Repeat steps (2. through (6., selecting a red box on the map to display another image. Note that when the box is selected, it turns yellow.

   If necessary, click the Roam icon in the Graphical Query viewer tool bar, and then drag the image in the Graphical Query viewer to manipulate the image view. (If you reset the display, be prepared to start back at step 2. on page 62.)

   NOTE: Magnified boxes that are deselected are red in color; selected boxes are yellow.

8. In the Image Catalog tool bar, click the Display Another icon to display the image you selected in step 7. in a second Viewer.

View Information

1. To view all of the information for the image in your first Viewer in one convenient form, highlight that file in the Image Catalog by clicking on the **Record** number of that file.

   The file is highlighted in yellow.

2. In the Image Catalog dialog, select **View | Form View** or click the Catalog Default icon .

   The Form View dialog opens, listing the characteristics of the file you selected.

3. In the bottom left corner of the Form View dialog, there are up and down arrows. Click twice on the down arrow, noting that the information in the Form View dialog corresponds with the highlighted file in the Image Catalog.
4. Click **Close** to close the Form View dialog.

5. Another way to view information on individual files is to use the ImageInfo dialog. Select **View | Image Info** from the Image Catalog or click the Info icon .

   The ImageInfo dialog opens.

6. Select **File | Close** from the ImageInfo dialog.

Modify Views

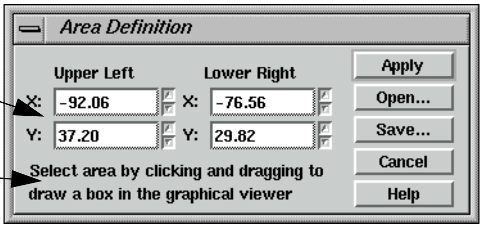
1. In the Graphical Query viewer, select **File | Area Definition**.

   The Area Definition dialog opens.

Enter the coordinates manually here

or

Follow these instructions here


Area Definition
Upper Left
Lower Right
Apply
Open...
Save...
Cancel
Help

X: -92.06 X: -76.56

Y: 37.20 Y: 29.82

Select area by clicking and dragging to draw a box in the graphical viewer

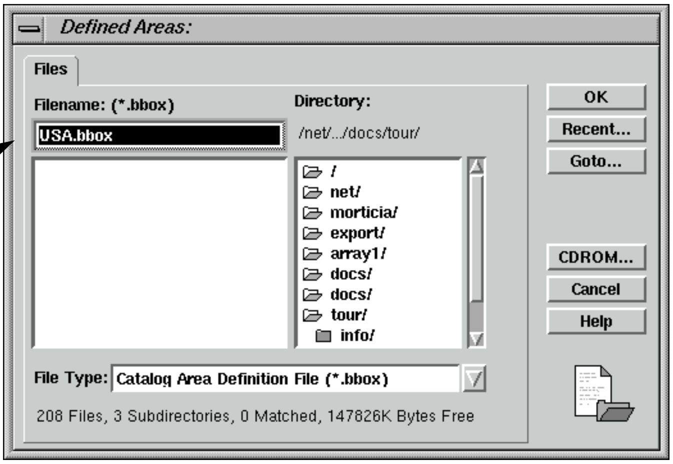
The Area Definition dialog lets you store the extent of particular areas of interest for quick retrieval. For example, if your study area covers the states of North and South Dakota, you could save a view of just that area. You can either enter the coordinates manually into the Area Definition dialog, or you can select an area by drawing a box in the Graphical Query viewer.

The upper left and lower right coordinates are the coordinates of the area currently viewed in the Graphical Query viewer. You can save these coordinates to a bounding box file (.bbox).

2. Click **Save** in the Area Definition dialog.

   The Defined Areas dialog opens.

Enter the directory and file name here


Defined Areas:
Files
OK
Recent...
Goto...
CDROM...
Cancel
Help
/
net/
morticia/
export/
array1/
docs/
docs/
tour/
info/

Filename: (*.bbox) USA.bbox

Directory: /net/.../docs/tour/

File Type: Catalog Area Definition File (*.bbox)

208 Files, 3 Subdirectories, 0 Matched, 147826K Bytes Free

3. In the Defined Areas dialog under **Filename**, enter the name **USA** in the directory of your choice and then click **OK**.

4. Click **Cancel** in the Area Definition dialog to close it.

5. In the Graphical Query viewer, click the Roam icon and then drag on the image to view a different area.

6. In order to recall the view you saved as **USA.bbox**, select **File | Area Definition** in the Graphical Query viewer, and then click **Open** in the Area Definition dialog.

   The Defined Areas dialog opens.

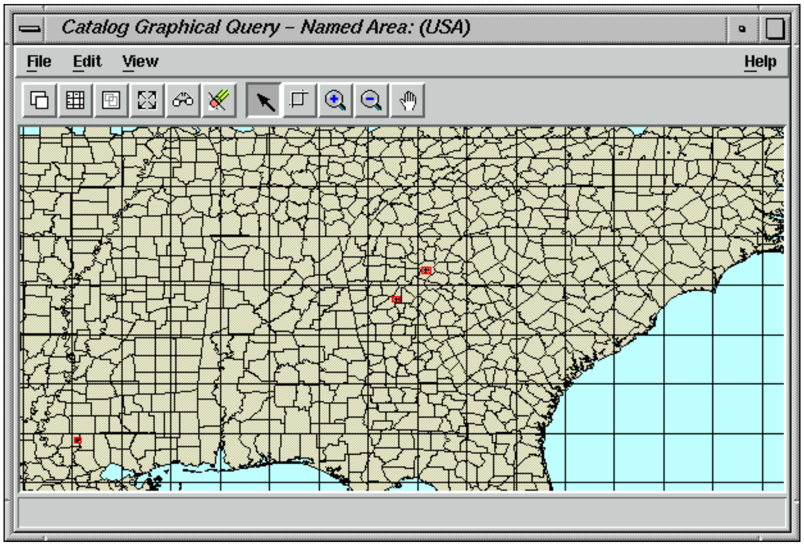
7. In the Defined Areas dialog under **Filename**, click on **USA.bbox**.

8. Click **OK** in the Defined Areas dialog.

   The Defined Areas dialog closes and the coordinates in the Area Definition dialog change.

9. Click **Apply** and then **Cancel** in the Area Definition dialog.

   The Graphical Query viewer redisplays the view you saved as **USA.bbox**.



Archive Data

The following steps detail how data are archived.

You must have a non-rewinding tape device already configured in order to archive data.

1. Select the image in the Viewer by clicking on its **Record** number in the Image Catalog.

   The file is highlighted in yellow.

2. In the Image Catalog, select **File | Archive**, or click on the Archive icon in the tool bar.

   The Archive dialog opens.

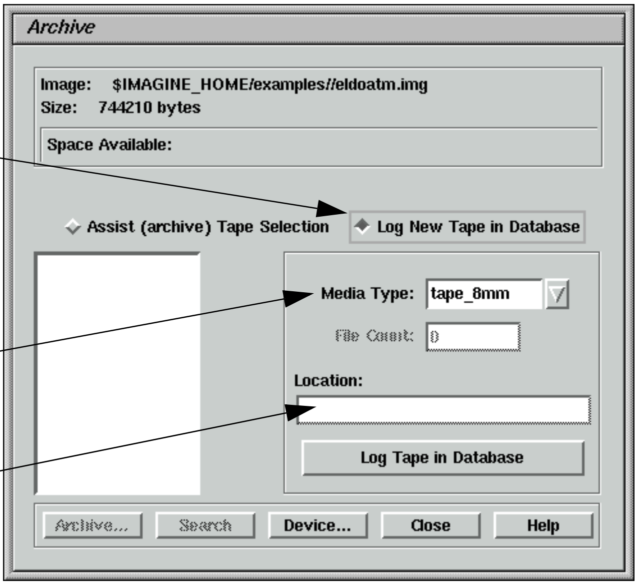


Image: $IMAGINE_HOME/examples//eldoatm.img
Size: 744210 bytes
Space Available:
Media Type: tape_8mm
File Count: 0
Location:

Click here to set up the tape

Click here to select the media type

Enter the location of the tape here

Follow steps (3. through (6. for initial setup only.

3. In the Archive dialog, click **Log New Tape in Database** to set up the tape to be used.

4. Click on the **Media Type** popup list and select either **tape_8mm** or **tape_exabyte**.

5. Enter the **Location** of the tape. This is simply a note to yourself, indicating where you store the tape or which tape you are using (e.g., top shelf or tape #16).

6. Click **Log Tape in Database** to add this tape to the list of available tapes. A message displays with the identification number that should be placed on the tape for later retrieval. The new tape number is displayed in the scrolling list of available tapes.

7. Click the tape from the list.

8. Click **Device** to select the actual tape device to which you archive. A list of configured tape devices displays.

9. In the Select Configured Devices dialog, click the device to which you archive. It must be configured as a non-rewinding device.

10. Click **Select** to select the device.

11. In the Archive dialog, click **Archive** to archive the selected image.

12. You are instructed to place the selected tape in the selected device. Click **Continue** when the tape is loaded. You are informed when the process is complete.

    NOTE: For subsequent images, omit steps (3. through (6.

CHAPTER THREE

Map Composer

Introduction

The ERDAS IMAGINE Map Composer is an editor for creating cartographic-quality maps and presentation graphics. Maps can include single or multiple continuous raster layers, thematic (GIS) layers, vector layers, and annotation layers.

Map Composer's extensive annotation capabilities allow you to automatically generate text, legends, scale bars, grid lines, tick marks, borders, symbols, and more. You can select from over 16 million colors, multiple line styles, and over 60 text fonts.

Approximate completion time for this tour guide is 40 minutes.

Create a Map

In this tutorial, you create a map using the file **modeler_output.img** in the <IMAGINE_HOME>/examples directory. This file contains SPOT panchromatic data overlaid with an environmental sensitivity analysis file.

In creating this map, you use these basic steps:

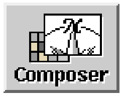
- plan the map
- start Map Composer
- prepare the data layers
- draw the map frame
- add a neatline and tick marks
- make scale bars
- create a legend
- add a map title
- place a north arrow
- write descriptive text
- print the map

This tour guide also contains information about editing map frames, deleting map frames, and editing map compositions.

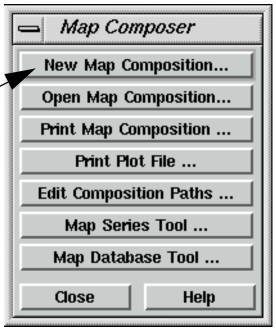
Start Map Composer

You must have ERDAS IMAGINE running.

1. Start Map Composer by clicking the Composer icon on the ERDAS IMAGINE icon panel.



The **Map Composer** menu displays.

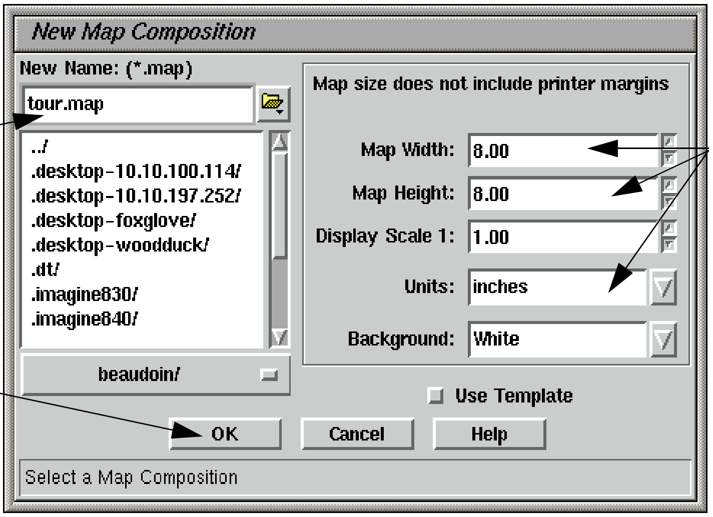

Map Composer
New Map Composition...
Open Map Composition...
Print Map Composition ...
Print Plot File ...
Edit Composition Paths ...
Map Series Tool ...
Map Database Tool ...
Close
Help

Click here to create a new map

2. In the **Map Composer** menu, select **New Map Composition** to create a new map composition.

You can also create a new composition by selecting **File | New | Composition** *from the Viewer menu bar.*

The New Map Composition dialog opens.


New Map Composition
../
.desktop-10.10.100.114/
.desktop-10.10.197.252/
.desktop-foxglove/
.desktop-woodduck/
.dt/
.imagine830/
.imagine840/
beaudoin/
Map size does not include printer margins
☐ Use Template
OK
Cancel
Help
Select a Map Composition

New Name: (*.map) tour.map

Map Width: 8.00

Map Height: 8.00

Display Scale 1: 1.00

Units: inches

Background: White

Type a name for your map here

Click here to make a new map

Specify map size here in meters or in inches

3. In the New Map Composition dialog under **New Name**, type a unique name for your map (such as **tour.map**). Be sure that you have write permission in the directory where you are creating the new map.

4. Drag across the value in **Map Width** to select it.

5. Type **7.5** and then press the Return key on your keyboard.

6. Drag across the value in **Map Height** to select it.

7. Type **10.0** and then press the Return key on your keyboard.

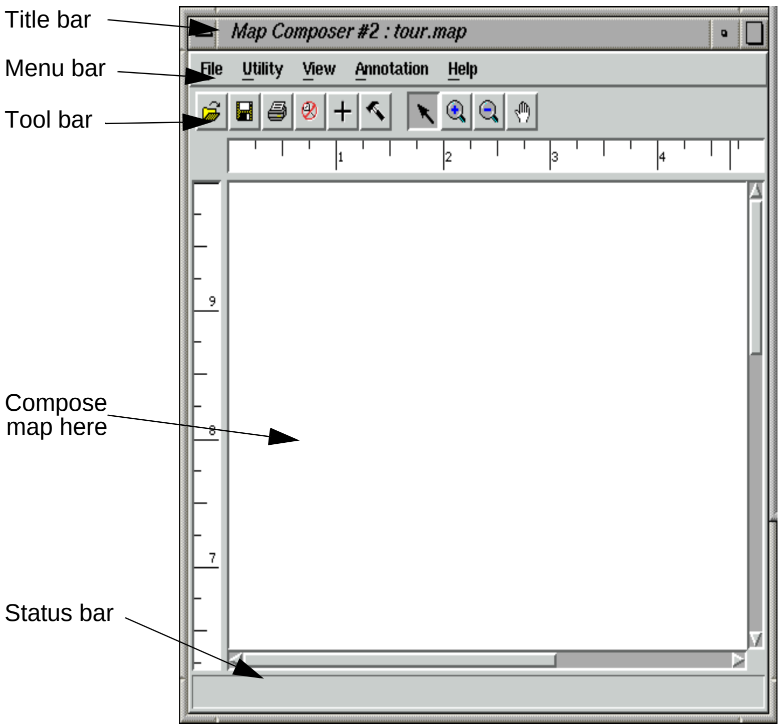
   Most printers have a small margin on all sides of a sheet that are unprintable. Therefore, even though you output your map to 8.5” × 11.0” paper, you must specify a smaller size here for the actual live area of the map. This ensures that no part of the map occupies that unprintable area, and it leaves a margin around the edge of the composition.

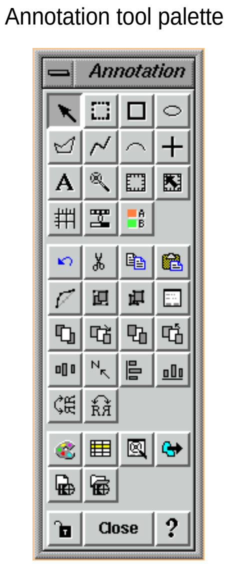
8. Accept the **Display Scale 1** of **1.00** and **inches** for the **Units**.

9. Click **OK** in the New Map Composition dialog.

   A blank Map Composer viewer displays along with the Annotation tool palette.

   NOTE: The tool palette that displays on your screen may look different from the following one, depending upon your ERDAS IMAGINE preferences.


Title bar
Menu bar
Tool bar
Compose map here
Status bar
Map Composer #2 : tour.map
File
Utility
View
Annotation
Help
1
2
3
4
9
8
7


Annotation tool palette
Annotation
Close
?

10. With your cursor in the Map Composer viewer, right-hold **Fit Map To Window** from the **Quick View** menu so that you can see the entire map composition page.

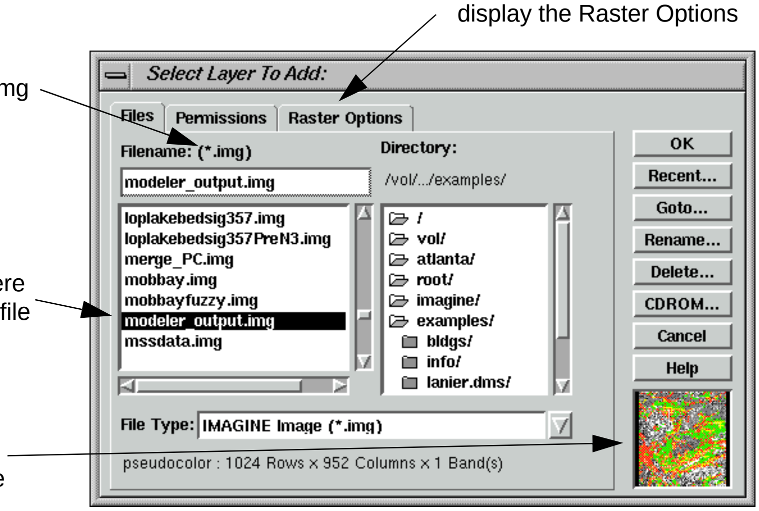
Prepare the Data Layers

You must determine what data layer(s) you are going to use in your composition and then display the layer(s) in a Viewer window. In this exercise, you use **modeler_output.img** from the <IMAGINE_HOME>/examples directory.

ERDAS IMAGINE must be running, and you must have a Viewer and a Map Composer viewer open.

1. In the Viewer, click the Open icon on the tool bar or select **File | Open | Raster Layer** from the menu bar.

   The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
Directory:
loplakebedsig357.img
loplakebedsig357PreN3.img
merge_PC.img
mobbay.img
mobbayfuzzy.img
modeler_output.img
mssdata.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) modeler_output.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

pseudocolor : 1024 Rows x 952 Columns x 1 Band(s)

Second, click here to display the Raster Options

The file type should be *.img

First, click here to select the file

A preview of the image displays here

2. In the Select Layer To Add dialog under **Filename**, click on **modeler_output.img**. Be sure that the source directory is <IMAGINE_HOME/examples> (the default).
3. Click on the **Raster Options** tab and then select the **Fit to Frame** option so that you can see the entire file and can more easily select the area to include in the map.
4. Click **OK** in the Select Layer To Add dialog.

Draw the Map Frame

You can now begin creating your map composition. The first step in creating your map is to define the map frame.

What is a Map Frame?

A map frame is drawn, resized, and selected like other annotation elements, but a map frame works like a Viewer. Map frames can contain raster, vector, and annotation layers that you want to include in your composition.

Although map frames display the layer(s) you want to include in a map composition, the data in a map frame are not copied, but are referenced. When you create a map frame, simply click in the Viewer where the layers you want to use in the map are displayed.

Map Frame Dimensions

There are three ways to select the dimensions of a map frame. You use different options depending on the image area you want in the final map composition. The dimensions of a map frame are expressed in the following ways:

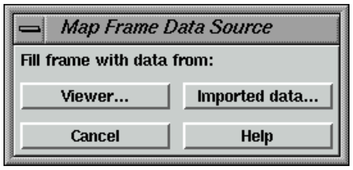
- The map area is the area in the Viewer that is displayed in the map frame in the map composition. It corresponds to the dimensions of the area on the ground in map units.
- The frame area is the area used by the map frame in the map composition. It is the area on the page occupied by a particular image. It is defined in page units.
- The scale (like the scale used in geometric correction) is the ratio of distance in the map frame to the distance that is represented on the ground. For example, you can define an area showing a scale of 1:24,000.

A Viewer must be open, with the data layer you want to use in your composition displayed in it. You must also have an open Map Composer viewer.

1. From the Annotation tool palette, click the Map Frame icon to draw the boundary of the map frame.
2. Near the top of the Map Composer viewer, Shift-drag your cursor downward at an angle to draw the map frame. You position and size the map frame later.

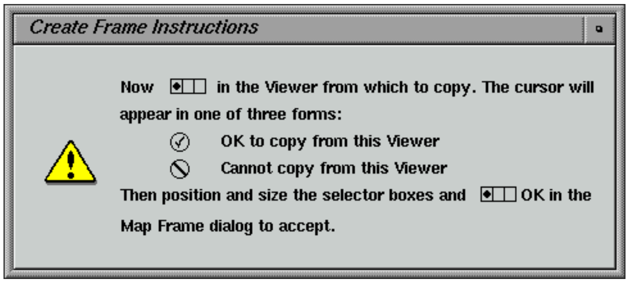
   NOTE: Pressing the Shift key while drawing the map frame allows you to draw a perfect square.

   When the mouse is released, the Map Frame Data Source dialog opens.



3. In the Map Frame Data Source dialog, click the **Viewer** button to select the source image from the Viewer.

   The Create Frame Instructions dialog displays.



Create Frame Instructions

Now ▣ in the Viewer from which to copy. The cursor will appear in one of three forms:

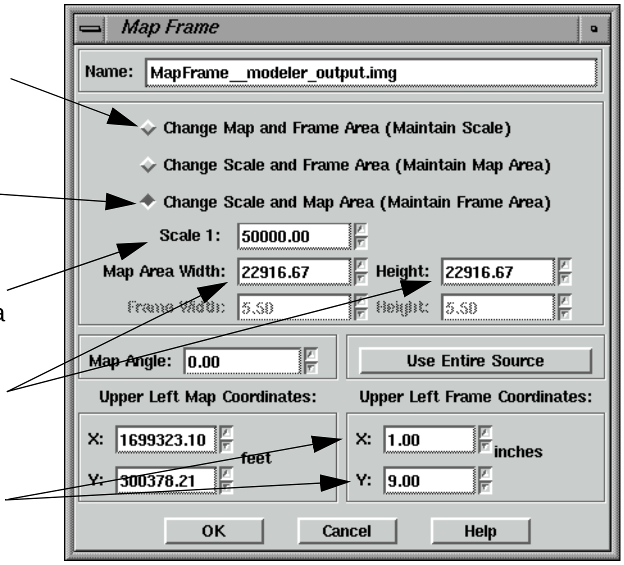
✓ OK to copy from this Viewer

⊘ Cannot copy from this Viewer

Then position and size the selector boxes and ▣ OK in the Map Frame dialog to accept.

4. Click anywhere in the image in the Viewer to reference the displayed image to the new map frame.

   The Map Frame dialog opens, giving you options for sizing, scaling, and positioning the map frame.



Map Frame

Name: MapFrame__modeler_output.img

◇ Change Map and Frame Area (Maintain Scale)

◇ Change Scale and Frame Area (Maintain Map Area)

◆ Change Scale and Map Area (Maintain Frame Area)

Scale 1: 50000.00

Map Area Width: 22916.67 Height: 22916.67

Frame Width: 5.50 Height: 5.50

Map Angle: 0.00

Upper Left Map Coordinates: X: 1699323.10 Y: 300378.21 feet

Upper Left Frame Coordinates: X: 1.00 Y: 9.00 inches

Click here to adjust the size of the map frame

Click here to adjust the position of the map frame

Click here to enter the scale of the data in the map frame

Click here to enter map frame height and width in inches

Click here to enter map frame X and Y coordinates

A cursor box also displays in the Viewer. This cursor box allows you to select the area you want to use in the map composition.

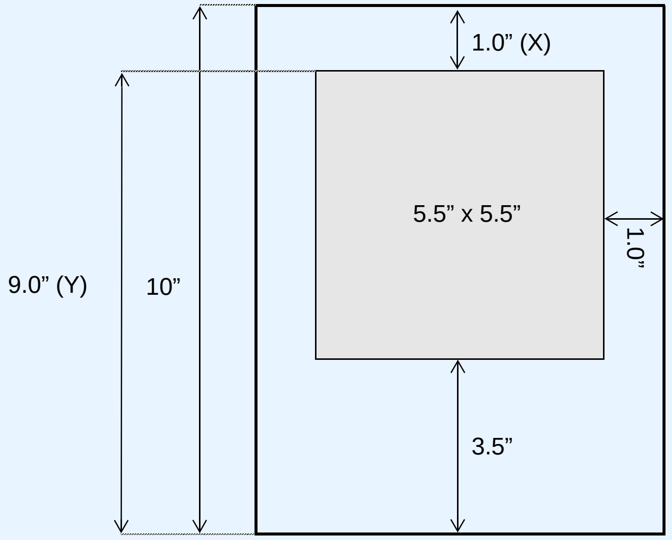
Adjust the Size of the Map Frame

You can move the map frame in the Map Composer viewer window and the cursor box in the Viewer by dragging and resizing them with the mouse, or you can move one or both boxes by manipulating the information in the Map Frame dialog. You can also rotate the box in the Viewer if you want to change the orientation.

Next, you move the map frame by setting parameters in the Map Frame dialog. Then you can select the image area you like by using the mouse to move the cursor box in the Viewer.

The three buttons at the top of the Map Frame dialog allow you to adjust two parameters while keeping the other frozen. Start by selecting the size of the map frame (**Change Scale and Frame Area**), and then freeze that option to select the scale and position of the map frame (**Change Scale and Map Area**).

Your map frame is positioned according to the following illustration:



The origin of the Map Composer viewer is the lower left corner.

1. In the Map Frame dialog, click the **Change Map and Frame Area (Maintain Scale)** option so that you can accurately size the map frame in the Map Composer viewer window.
2. In the Map Frame dialog, double-click on the value in **Frame Width** to select it.
3. Type **5.5** for the width of the map frame and then press the Return key on your keyboard.
4. Double-click on the value in the frame **Height** (to the right of **Frame Width**) to select it.
5. Type **5.5** for the height of the map frame and then press the Return key on your keyboard.

Adjust the Position of the Map Frame

1. In the Map Frame dialog, click the **Change Scale and Map Area (Maintain Frame Area)** option so that you can select a map area in the Viewer without losing the dimensions of the map frame in the Map Composer viewer.

2. In the Map Frame dialog, under **Upper Left Frame Coordinates**, change the **Y** value to **9.0**. Press Return on your keyboard.

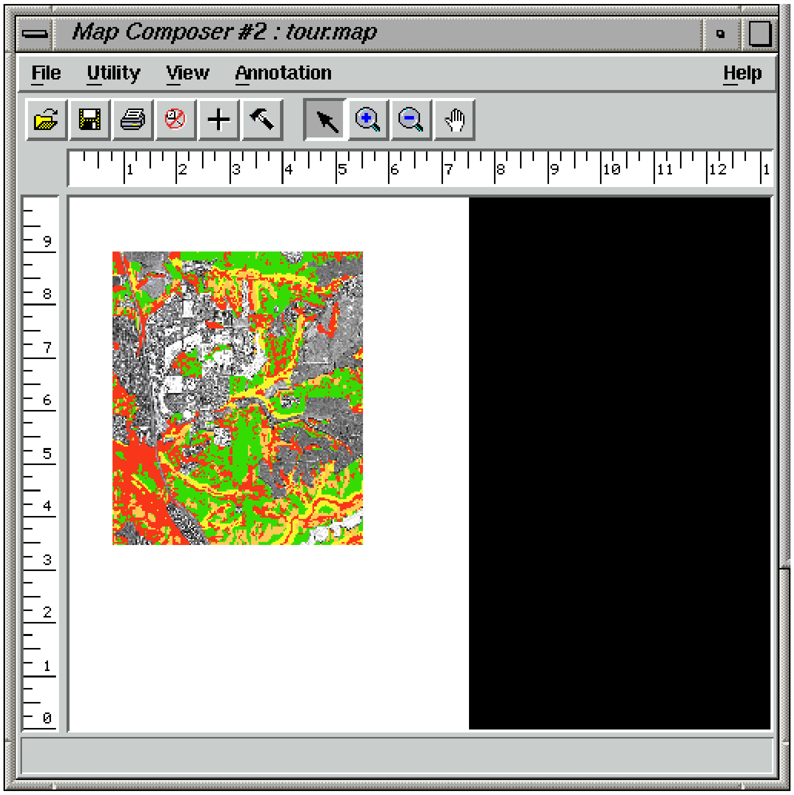
3. Double-click on the value in **Scale 1** to select it. Type **50000**. Press Return on your keyboard.

4. With your cursor inside the cursor box in the Viewer, drag the cursor box to the area you want to display in the map composition.

5. When the cursor box is positioned to your satisfaction, click **OK** in the Map Frame dialog to reference that portion of the image to the map composition.

   The image area that you selected in the Viewer is now displayed in the map frame in the Map Composer viewer.

6. To enlarge the image in the Map Composer viewer, drag on the corners of the Map Composer viewer to enlarge it, and the right-hold to select **Fit Map To Window** from the **Quick View** menu.



7. You are now finished with the Viewer, so select **File | Close** from the Viewer menu bar to close it.

Choose Your Path

- If you are satisfied with your map frame and the image area you have selected, proceed to "Add a Neatline and Tick Marks" on page 78.
- If you want to edit your map frame or change the image area you selected, proceed to "Edit the Map Frame".

Edit the Map Frame

The Map Composer is very flexible in allowing you to place a map frame in a composition based on scale, image area, etc. Once you have placed a map frame in a composition, you can move it, change the size of it, and change the image area within it.

If you want to change the image you are using, you must delete the map frame and redraw it or edit the .map file.

Choose Your Path

- To edit the map frame, proceed with this section.
- To delete the map frame, proceed to "Delete the Map Frame" on page 77.
- To edit the .map file, proceed to "Edit Composition Paths" on page 92.

1. To edit a map frame, click the Select Map Frame icon in the Annotation tool palette.
2. Click in the map frame you want to edit to select it.
3. In the Map Composer viewer, select **Annotation | Element Properties** or double-click in the map frame.

   A new Viewer opens, with the image you are using displayed. A white cursor box indicates the area currently in the map frame. The Map Frame dialog also opens, displaying the settings you originally entered to position and size the map frame.

4. Change the information in the Map Frame dialog and/or move the cursor box in the Viewer.
5. Click **OK** in the Map Frame dialog when you are satisfied with the map frame.
6. Select **File | Close** from the Viewer menu bar.

Delete the Map Frame

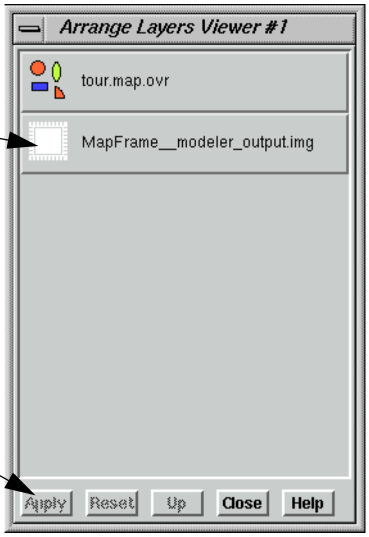
If you want to delete the map frame altogether, follow the next series of steps.

Deleting a map frame cannot be undone.

You must have your map composition open.

1. In the Map Composer viewer menu bar, select **View | Arrange Layers**.

The Arrange Layers dialog opens.



Press the third mouse button to delete this map frame

Click here to delete the map frame

2. In the Arrange Layers dialog, move your pointer to the box titled **MapFrame__modeler_output.img** and right-hold to select **Delete Layer** from the Frame Options popup list.

3. In the Arrange Layers dialog, click **Apply** to delete the map frame.

4. Click **Close** in the Arrange Layers dialog.

You are now ready to redraw the map frame. Return to "Draw the Map Frame" on page 72.

Add a Neatline and Tick Marks

Now, add a neatline and labeled tick marks around the image in the map composition.

Neatlines and Tick Marks

- A neatline is a rectangular border around a map frame.
- Tick marks are small lines along the edge of the map frame or neatline that indicate regular intervals of distance. Tick marks are usually labeled in meters, feet, or other units.

The Map Composer allows you to generate a neatline and labeled tick marks at the same time. A set of these elements for a map frame is actually a group of line and text elements that is automatically generated to your specifications. (You can also generate grid lines in the same step, but grid lines are not included in this map.)

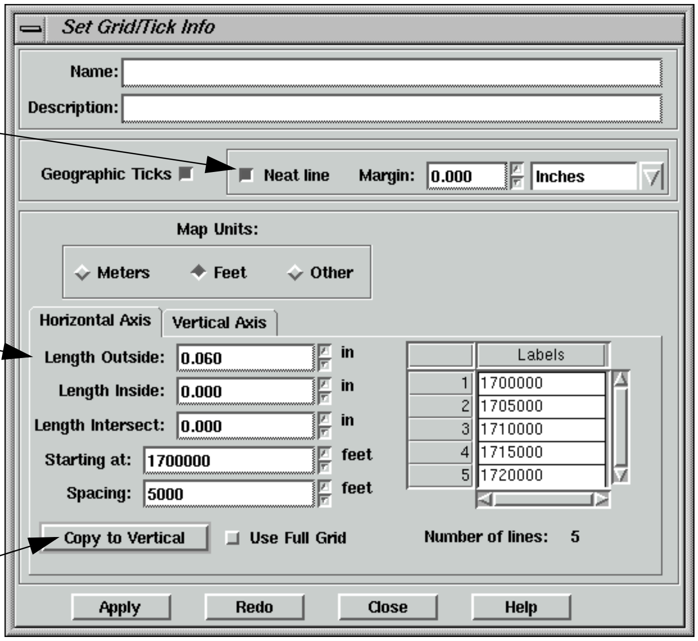
You must have a Map Composer viewer open containing a map frame referenced to a georeferenced image in order to generate georeferenced tick marks. The Annotation tool palette must also be open.

1. If you have not already done so, right-hold **Fit Map To Window** in the Map Composer viewer so that you can see the entire map composition page.

2. Click the Grid/Tick icon on the Annotation tool palette.

3. Click on the image inside the map frame on which you want to place the neatline and tick marks.

   The Set Grid/Tick Info dialog opens.



Name:

Description:

Geographic Ticks ☑ ☑ Neat line Margin: 0.000 Inches

Map Units: ○ Meters ● Feet ○ Other

Horizontal Axis | Vertical Axis

Length Outside: 0.060 in

Length Inside: 0.000 in

Length Intersect: 0.000 in

Starting at: 1700000 feet

Spacing: 5000 feet

	Labels
1	1700000
2	1705000
3	1710000
4	1715000
5	1720000

☐ Use Full Grid

Number of lines: 5

Specify neatline here

Enter tick mark information here for the horizontal axis

Click here to copy horizontal tick information to the vertical axis

4. Accept the default of **Neat line** to put a neatline around the map and leave the **Margin** at **0** so that the neatline fits to the edge of the map frame.

5. In the **Horizontal Axis** options, drag across the **Length Outside** field to select it.

6. Type a tick length of **0.06**. Press Return on your keyboard.

   Tick marks extend 0.06” outside of the map frame.

7. Drag across the **Spacing** field to select it.

8. Type **5000**. Press Return on your keyboard.

   The **Number of lines** is about **4**, indicating that there are 4 horizontal tick marks (depending on the actual image area you selected).

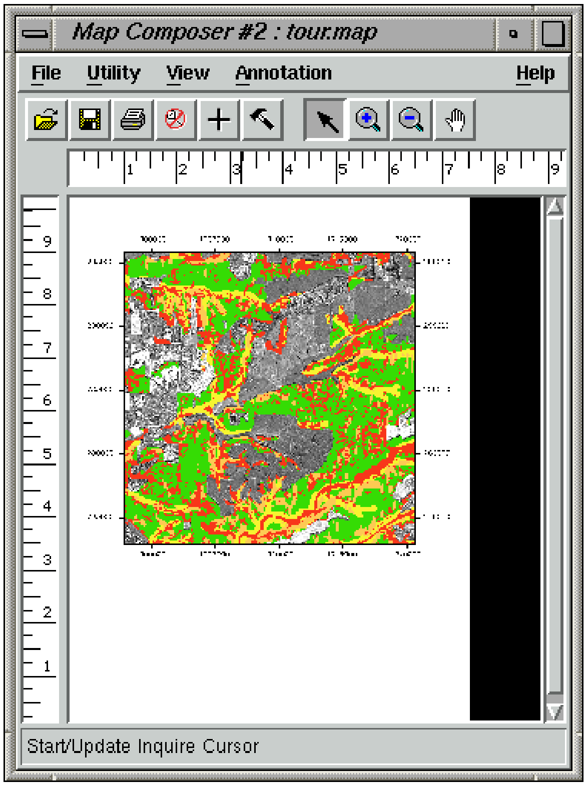
9. Click the **Copy to Vertical** option to apply these settings to the vertical axis.

   The same settings are applied to the vertical axis.

10. Click the **Vertical Axis** tab to verify neatline and label information for the vertical axis.

11. Click **Apply** in the Set Grid/Tick Info dialog to place the neatline and tick marks on the map.

12. If you are satisfied with the appearance of the neatline, click **Close** in the Set Grid/Tick Info dialog. Otherwise, you may make adjustments in the Set Grid/Tick Info dialog and click **Redo** to apply them. Your map should look similar to the following:



Start/Update Inquire Cursor

Change Text/Line Styles

The text and line styles used for neatlines, tick marks, and grid lines depend on the default settings in the Styles dialog. You can either set the styles before adding this annotation to your map, or you can change the styles once they are placed in the map.

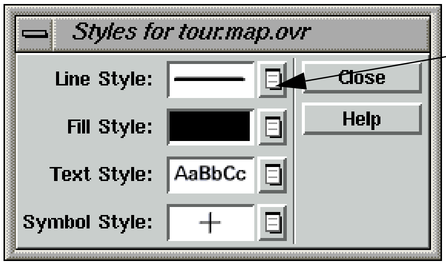
Next, set the line style to 1 point for the neatline and tick marks, and the text size to 10 points for the tick labels.

1. Select the group of ticks, tick labels, and the neatline by clicking on any of the number labels outside of the map frame.

   A selection box displays around the entire group.

2. From the Map Composer viewer menu bar, select **Annotation | Styles**.

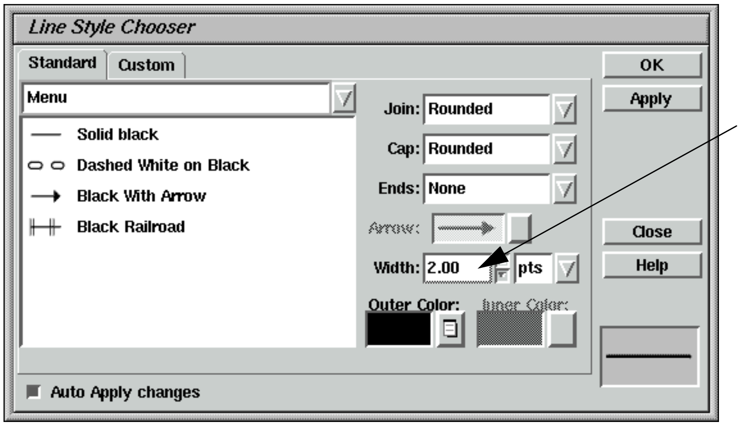
   The Styles dialog opens.


Styles for tour.map.ovr
Line Style:
Fill Style:
Text Style:
AaBbCc
Symbol Style:
Close
Help

Hold on this popup list to select Other

3. In the Styles dialog, hold on the popup list next to **Line Style** and select **Other**.

   The Line Style Chooser dialog opens.


Line Style Chooser
Standard
Custom
Menu
Solid black
Dashed White on Black
Black With Arrow
Black Railroad
Arrow:
Outer Color:
Inner Color:
OK
Apply
Close
Help
Auto Apply changes

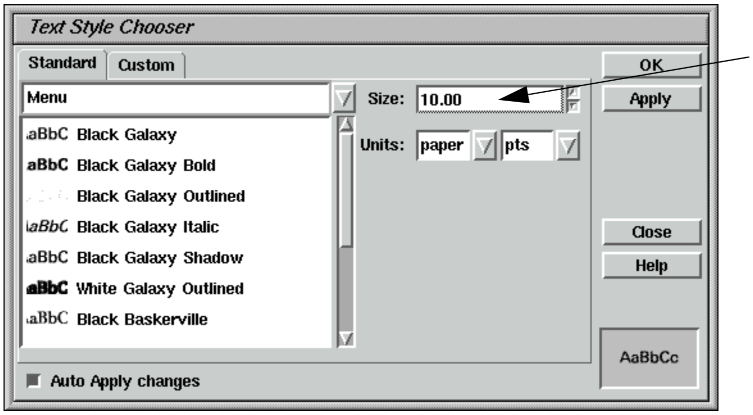
Join: Rounded
Cap: Rounded
Ends: None
Width: 2.00 pts

Change the line width here

4. In the Line Style Chooser dialog next to **Width**, enter **1.00** to change the width in points.
5. Click **Apply** and then **Close** in the Line Style Chooser dialog.

   The group redraws with the new line width.
6. In the Styles dialog, hold on the popup list next to **Text Style** and select **Other**.

   The Text Style Chooser dialog opens.


Text Style Chooser
Standard
Custom
Menu
aBbC Black Galaxy
aBbC Black Galaxy Bold
Black Galaxy Outlined
aBbC Black Galaxy Italic
aBbC Black Galaxy Shadow
aBbC White Galaxy Outlined
aBbC Black Baskerville
OK
Apply
Close
Help
AaBbCc
Auto Apply changes

Size: 10.00

Units: paper pts

Verify that the text size is 10 points

7. In the Text Style Chooser dialog, verify that the **Size** is **10.00**.

8. Click **Apply** and then **Close** in the Text Style Chooser dialog.

9. Deselect this annotation group by clicking anywhere in the map composition window outside of the selection box.

10. Click **Close** in the Styles dialog.

Make Scale Bars

A scale bar indicates the scale of the image on the map. You can create one scale bar, or several, showing the scale in different units. A scale bar is actually a group of elements that is automatically generated to your specifications.

In this section, place two scale bars in your map composition, showing scale in kilometers and miles. Then center them under the map frame.

You can create scale bars only for map frames containing georeferenced data.

You must have a map composition open and it must contain georeferenced data. The Annotation tool palette must also be open.

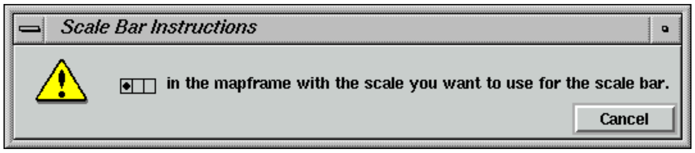
1. To place scale bars, select the Scale Bar tool from the Annotation tool palette.

2. Move the cursor into the Map Composer viewer and the cursor changes to the scale bar positioning cursor.

3. Drag the mouse to draw a box under the right corner of the map frame in the Map Composer viewer, outlining the length and location of the scale bar(s).

   You can change the size (length) and location later, if needed.

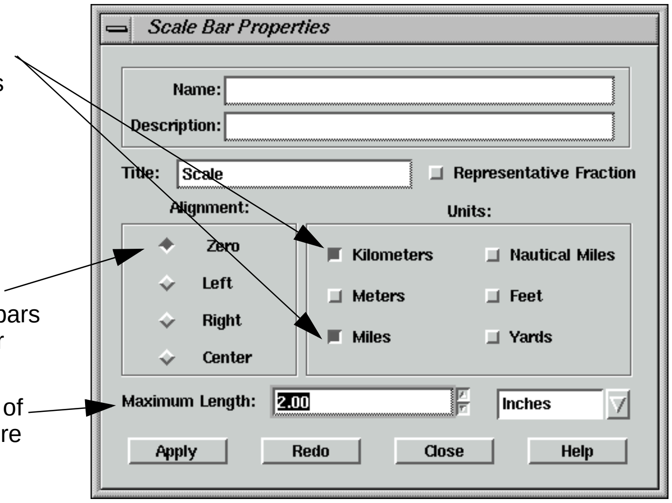
   When you release the mouse button, the Scale Bar Instructions dialog is activated.



in the mapframe with the scale you want to use for the scale bar.

4. Follow the instructions in the Scale Bar Instructions dialog by clicking in the map frame to indicate that this is the image's scale you are showing.

   The Scale Bar Properties dialog opens.



Title: Scale

☐ Representative Fraction

Alignment: ● Zero ○ Left ○ Right ○ Center

Units: ☒ Kilometers ☐ Nautical Miles ☐ Meters ☐ Feet ☒ Miles ☐ Yards

Maximum Length: 2.00 Inches

Click here and here to select units

Click here to center scale bars on each other

Select length of scale bars here

5. In the Scale Bar Properties dialog under **Units**, select **Kilometers** and **Miles** by clicking the appropriate checkboxes.

6. Set the **Maximum Length** to **2.0** inches. Press Return on your keyboard.

7. Click **Apply** in the Scale Bar Properties dialog.

   The scale bars display where you drew the box in the Map Composer viewer.

8. If you are satisfied with the appearance of the scale bar, click **Close** in the Scale Bar Properties dialog. Otherwise, you may make adjustments in the Scale Bar Properties dialog and click **Redo** to apply them.

Reposition Scale Bars

1. If you need to move the scale bars, first select them by clicking on one of the scale bars. To move, simply drag the selection box to the desired position. Remember to click outside the selection box to deselect the scale bars.

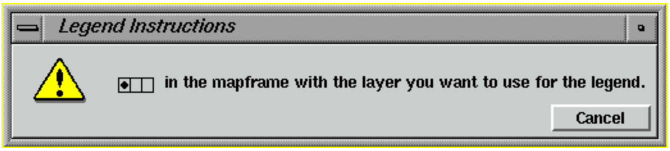
Create a Legend

A legend is a key to the colors that are used in a map. Legends created in the Map Composer are actually groups of elements that are generated automatically to your specifications.

Next, you create a legend explaining the four colors used in the overlaid environmental sensitivity analysis.

You must have a map composition open that contains thematic data. The Annotation tool palette should also be open.

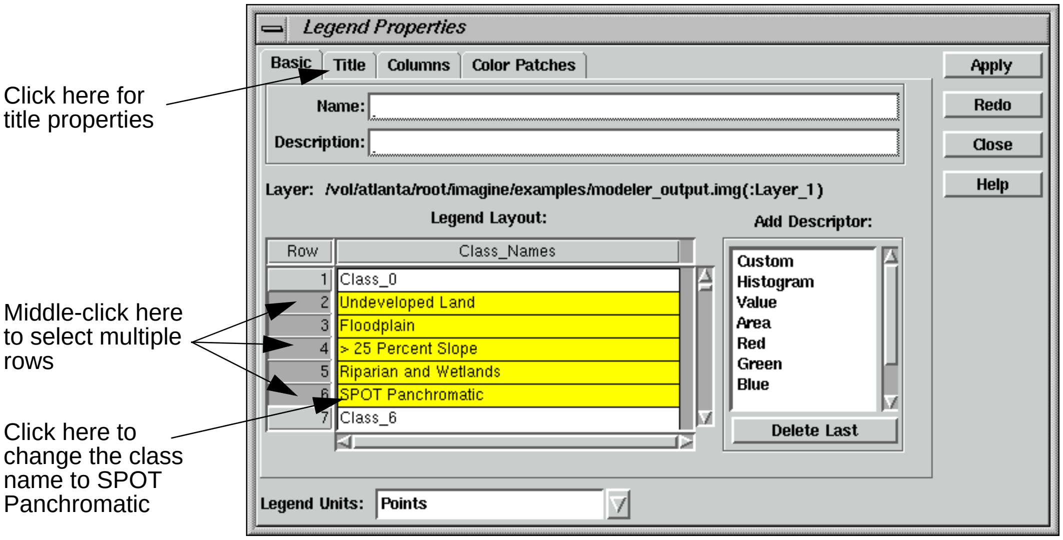
1. Click on the Legend icon in the Annotation tool palette.
2. Move the cursor into the Map Composer viewer and the cursor changes to the legend positioning cursor.
3. Click in the Map Composer viewer under the left side of the map frame to indicate the position of the upper left corner of the legend.
4. Click in the map frame to indicate that this is the image you want to use to create the legend. You are reminded to do this with the following dialog:


Legend Instructions

in the mapframe with the layer you want to use for the legend.

Cancel

The Legend Properties dialog opens, with the **Basic** properties displayed. The class names are listed under **Legend Layout**.


Legend Properties
Basic
Title
Columns
Color Patches
Apply
Redo
Close
Help
Name:
Description:
Legend Layout:
Add Descriptor:
Custom
Histogram
Value
Area
Red
Green
Blue
Delete Last

Layer: /vol/atlanta/root/imagine/examples/modeler_output.img(:Layer_1)

Row	Class_Names
1	Class_0
2	Undeveloped Land
3	Floodplain
4	> 25 Percent Slope
5	Riparian and Wetlands
6	SPOT Panchromatic
7	Class_6

Legend Units: Points

Click here for title properties

Middle-click here to select multiple rows

Click here to change the class name to SPOT Panchromatic

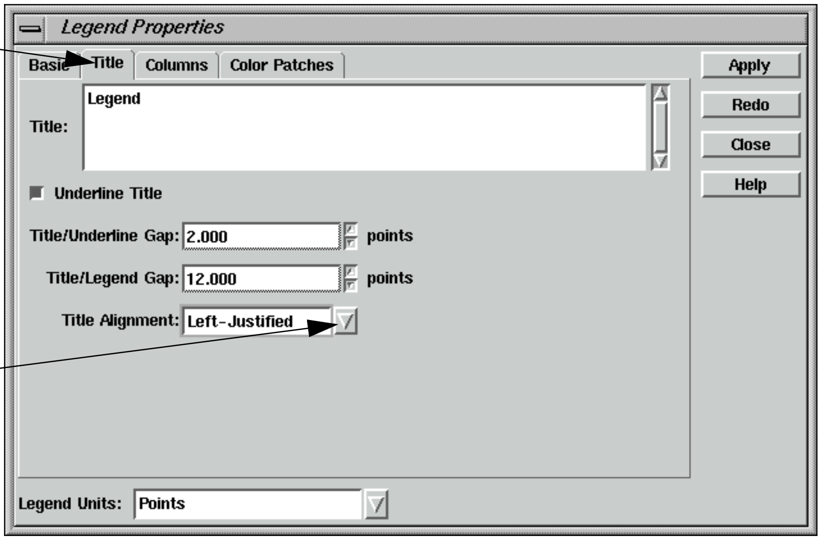
5. Under **Legend Layout**, click in the **Class_Names** field (entitled **Class_5**) of **Row** number **6**.

6. Type **SPOT Panchromatic**. Press Return on your keyboard.

7. Under **Legend Layout**, move your cursor to the **Row** column and click on **Row 2**, then middle-click on **Row 6** to select the classes to display in the legend.

   Rows **2** and **6**, and the rows in between them are highlighted in yellow. These are the only entries that are used in the legend.

8. In the Legend Properties dialog, click the **Title** tab at the top of the dialog.



Title: Legend

Title/Underline Gap: 2.000 points

Title/Legend Gap: 12.000 points

Title Alignment: Left-Justified

Legend Units: Points

Click this tab to view title properties

Click on this popup list to specify alignment

9. Click on the **Title Alignment** popup menu and select **Left-Justified**.

10. In the Legend Properties dialog, click **Apply**.

    The legend is drawn in the Map Composer viewer.

11. If you are not satisfied with the appearance of your legend, you may make adjustments in the Legend Properties dialog and click **Apply** or **Redo** to apply them.

12. Click **Close** in the Legend Properties dialog when finished.

Reposition Legend

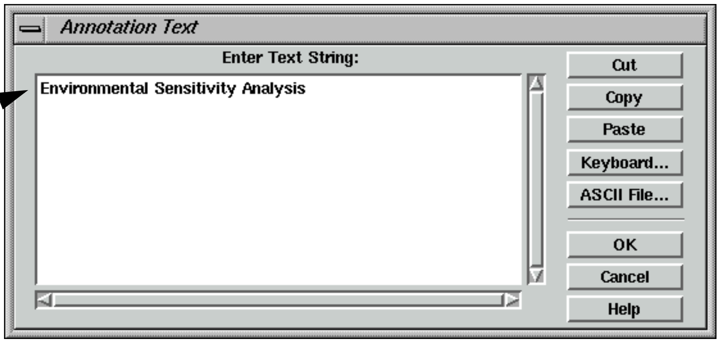
1. If you wish to reposition the legend, click on any of the color patches or text strings in the legend to select it. To move, hold and drag the selection box to the desired position. Remember to click outside the selection box to deselect the legend.

Add a Map Title

You must have a map composition and the Annotation tool palette open.

1. Click on the Text icon in the Annotation tool palette.
2. Move your cursor to the top of the map in the Map Composer viewer. The cursor becomes an I-beam , indicating that you are placing text.
3. Click where you want to place the text. The spot where you click is the bottom left corner of your text string.

   The Annotation Text dialog opens.



Environmental Sensitivity Analysis

Type text string here

4. Move your pointer into the **Enter Text String** area in the Annotation Text dialog.
5. Type **Environmental Sensitivity Analysis** in the text field.
6. Click **OK** in the Annotation Text dialog to place the text in the map composition.

   The text string is now displayed in the map composition.

Change Text Style

1. Click on the text string in the Map Composer viewer to select it.
2. From the Map Composer menu bar, select **Annotation | Styles**.

   The Styles dialog opens.

3. In the Styles dialog, hold on the popup list next to **Text Style** and select **Other**.

   The Text Style Chooser dialog opens.

4. In the Text Style Chooser dialog, change the text **Size** to **20** points.

   The preview window at the bottom right corner of the Text Style Chooser dialog illustrates the change in point size.

5. In the Text Style Chooser dialog, click on the **Custom** tab.

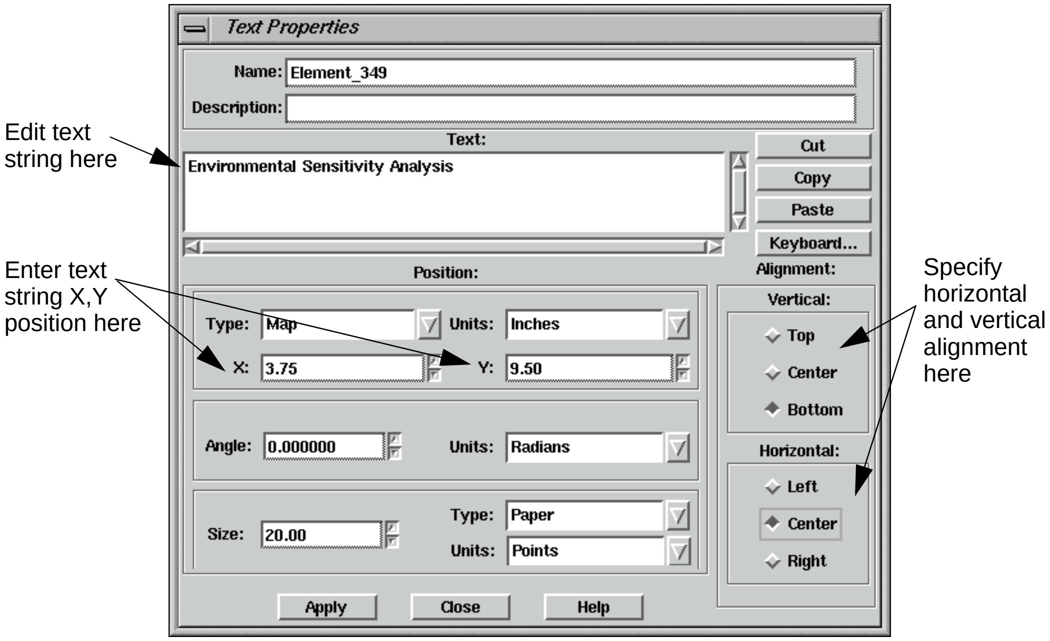
6. In the scrolling list of font names, scroll to the top of the list and select **Antique-Olive**. The preview window at the bottom right corner of the dialog illustrates the selected font.

7. Click **Apply** in the Text Style Chooser dialog to change the selected text in the map composition.

8. Click **Close** in the Text Style Chooser dialog if you are satisfied with your changes.

Position Text

1. Double-click on the text string you just edited. The Text Properties dialog opens, which allows you to edit, position, and align the text.



Text Properties

Name: Element_349

Description:

Text: Environmental Sensitivity Analysis

Position:

Type: Map Units: Inches

X: 3.75 Y: 9.50

Angle: 0.000000 Units: Radians

Size: 20.00 Type: Paper Units: Points

Alignment:

Vertical: ◇ Top ◇ Center ◆ Bottom

Horizontal: ◇ Left ◆ Center ◇ Right

Edit text string here

Enter text string X,Y position here

Specify horizontal and vertical alignment here

2. In the Text Properties dialog, under **Position**, drag across the **X** value to select it.

3. Type **7.5/2** to calculate the center of the map. Press Return. The value **3.75** is returned.

4. Change the **Y** value to **9.5** and press Return on the keyboard.

5. Under **Alignment**, the **Vertical** default should be **Bottom**. Click the **Center** radio button under **Horizontal**. This indicates that position **3.75** × **9.5** (that you just entered) is to be the bottom center of the text string.

6. Click **Apply** in the Text Properties dialog to center the text.

7. Click **Close** in the Text Properties dialog.

8. Deselect the text by clicking elsewhere in the background of the Map Composer viewer.

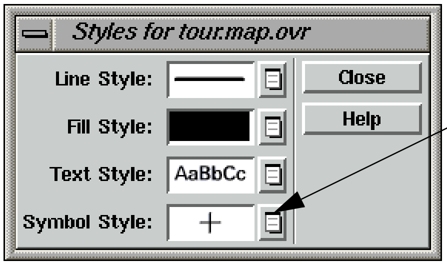
Place a North Arrow

Map Composer contains many symbols, including north arrows. These symbols are predrawn groups of elements that are stored in a library. Other symbols include school, church, marsh, landmark, and many others.

You must have a map composition and the Annotation tool palette open.

1. If the Styles dialog is not currently open, select **Annotation | Styles** from the Map Composer menu bar.

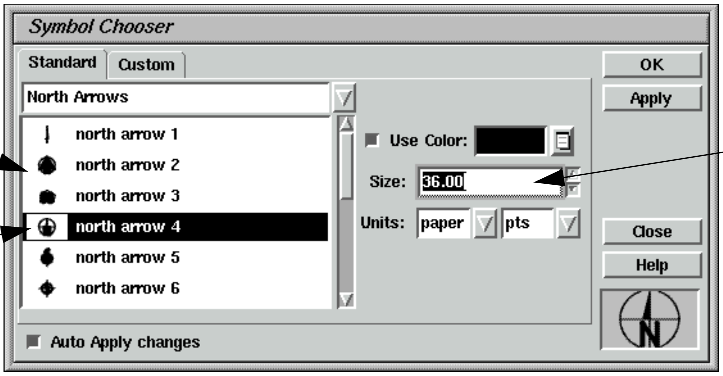
   The Styles dialog opens.


Styles for tour.map.ovr
Line Style:
Fill Style:
Text Style:
AaBbCc
Symbol Style:
Close
Help

Hold on this popup list to select Other

2. In the Styles dialog, hold the popup list next to **Symbol Style** and select **Other**.

   The Symbol Chooser dialog opens.


Symbol Chooser
Standard
Custom
North Arrows
north arrow 1
north arrow 2
north arrow 3
north arrow 4
north arrow 5
north arrow 6
Use Color:
Size:
36.00
Units:
paper
pts
OK
Apply
Close
Help
Auto Apply changes

Click here to select from a list of north arrows

Select this north arrow

Change the size here

3. In the Symbol Chooser dialog, click on the popup list and select **North Arrows**.

4. Select **north arrow 4** from the **North Arrows** list.

   The preview window at the bottom right corner of the Symbol Chooser dialog displays **north arrow 4**.

5. In the Symbol Chooser dialog, change the **Size** to **36** points (a size of 72 points is equal to one inch), and press Return on the keyboard.

   The preview window at the bottom right corner of the Symbol Chooser dialog displays the north arrow as it looks in the map composition.

6. Click **Apply** and then **Close** in the Symbol Chooser dialog to make this the default symbol.

   Note that the **North Arrow** is now the default symbol for **Symbol Style** in the Styles dialog.

7. Select the Symbol tool + from the Annotation tool palette.

8. In the Map Composer viewer, click under the map image, between the legend and the scale bars.

   The north arrow is placed on your composition. You can reposition it by clicking on it to select it, then dragging it to the new position.

   You can also double-click on a selected symbol to bring up the Symbol Properties dialog. Here, you can enter size and positioning measurements.

Write Descriptive Text

You can add descriptive text to your map to provide more information. The steps below include the instructions for adding two lines of text. However, you may add more if you like.

You must have a map composition and the Annotation tool palette open.

1. If the Styles dialog is not currently open, select **Annotation | Styles** from the Map Composer menu bar.

   The Styles dialog opens.

2. In the Styles dialog, hold on the **Text Style** popup list and select **Other**.

   The Text Style Chooser dialog opens.

3. In the Text Style Chooser dialog, change the text **Size** to **10** points.

4. Click on the **Custom** tab at the top of the Text Style Chooser dialog.

5. Check to be sure that **Fill Style** is set to **solid black**.

6. Click **Apply** to change the defaults.

7. Click **Close** in the Text Style Chooser dialog and in the Styles dialog.

Place Text

1. Click on the Text icon A in the Annotation tool palette to use the text option to write descriptive text.

2. Click in the bottom right side of the map composition to indicate where you want to place the text.

   The Annotation Text dialog opens.

3. Move your pointer into the Annotation Text dialog and type the following lines under **Enter Text String**. At the end of the first line, press Return to left-align the text.

San Diego, California<return>
Environmental Sensitivity Analysis

You may click the **ASCII File** *button to import the text from an existing ASCII text file.*

4. Click **OK** in the Annotation Text dialog to place the text.

The Annotation Text dialog automatically closes.

Save the Map Composition

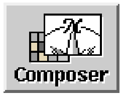
1. Save your map composition by clicking the Save icon in the Map Composer tool bar.

You can also save a composition by selecting **File | Save | Map Composition** *from the Map Composer menu bar.*

Print the Map Composition

ERDAS IMAGINE supports many output devices, including electrostatic plotters, continuous tone color printers, and PostScript devices. The printers you can use vary depending on your system configuration. These steps illustrate how to print to a PostScript printer.

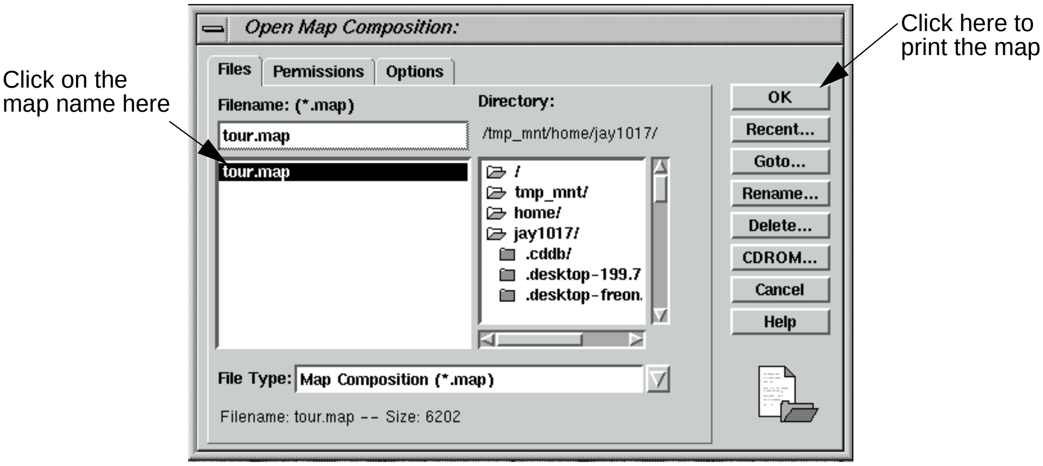
1. Click on the Composer icon from the ERDAS IMAGINE icon panel.



The **Map Composer** menu displays.

2. Select **Print Map Composition** from the **Map Composer** menu, or select **File | Print** from the Map Composer viewer menu bar.

The Compositions dialog opens.



Click on the map name here

Click here to print the map

Open Map Composition:

Filename: (*.map) tour.map

Directory: /tmp_mnt/home/jay1017/

File Type: Map Composition (*.map)

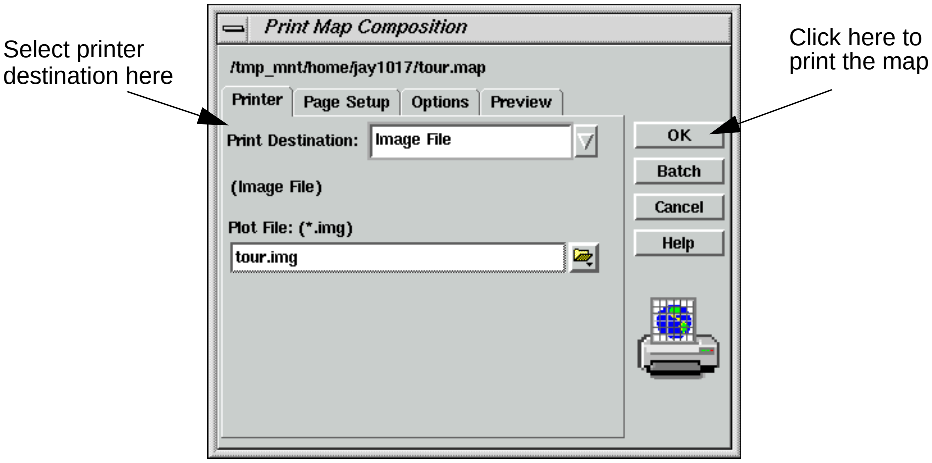
Filename: tour.map -- Size: 6202

Files
Permissions
Options
tour.map
/
tmp_mnt/
home/
jay1017/
.cddb/
.desktop-199.7
.desktop-freon.
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

3. Under **Filename**, click on the name of the map you previously created.

4. Click **OK** in the Compositions dialog.

   The Print Map Composition dialog opens.



Select printer destination here

Click here to print the map

Print Map Composition

/tmp_mnt/home/jay1017/tour.map

Print Destination: Image File

(Image File)

Plot File: (*.img) tour.img

Printer
Page Setup
Options
Preview
OK
Batch
Cancel
Help

5. Click on the **Print Destination** popup list to select the printer you want to use.

   If you do not have any output devices configured, you can output your map to an ERDAS IMAGINE image file (.img extension), and display it in a Viewer. When displaying maps converted to image format, assign bands 1, 2, 3 to R, G, B, respectively, in the Select Layer To Add dialog. This gives you an idea of what the map looks like if it is printed.

6. For this exercise, select **EPS File** in the **Print Destination** popup list to create an encapsulated PostScript file. This file can be sent to a PostScript printer using the standard file print command for your platform.

7. Click **OK** to print the map composition.

Edit Composition Paths

This section describes how to edit a map composition when you want to use another image in an existing map frame, or if the original image you used has been moved to another directory.

.map File

When you create a map or a graphic using Map Composer, a file is created with the .map extension. This file contains all of the specifications for your composition, such as size, position of the image, name of the image, annotation, etc. When you display or print a map composition, the software reads this .map file and recreates the map you originally composed.

So, although you place an image in a composition, you are actually only referencing it. The name of the image you are using is listed in the .map file. Therefore, when that image is enhanced or changed in any way, the image in the map composition also changes because it is the same image.

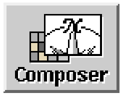
It is necessary to edit a .map file if you wish to move an image that has been used in a map composition to a new directory.

Editing Annotation

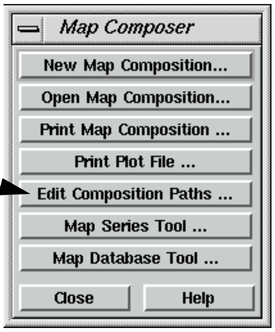
The annotation in a map composition can be edited interactively with the Annotation tools in the Annotation tool palette, using the same methods you used to place the annotation when

You must have a saved map composition (.map file).

1. Click the Composer icon on the ERDAS IMAGINE icon panel.



The **Map Composer** menu displays.


Map Composer
New Map Composition...
Open Map Composition...
Print Map Composition ...
Print Plot File ...
Edit Composition Paths ...
Map Series Tool ...
Map Database Tool ...
Close
Help

Click here to edit the composition

2. In the **Map Composer** menu, select **Edit Composition Paths**.

   The Map Path Editor displays.

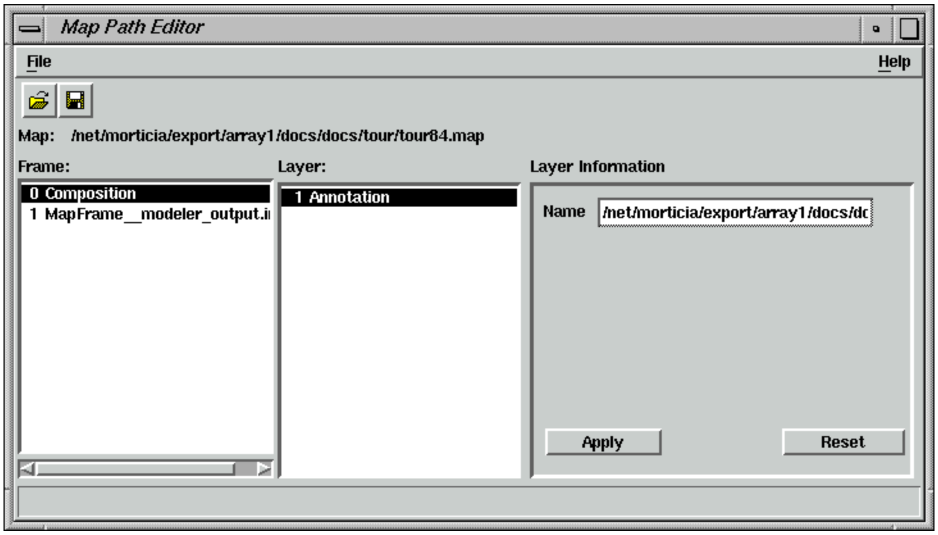
3. In the Map Path Editor, select **File | Open** or click the Open icon .

   The Compositions dialog opens.

4. In the Compositions dialog under **Filename**, select the map file you wish to edit (e.g., **tour.map**).

5. Click **OK** in the Compositions dialog.

   The information for the selected map file displays in the Map Path Editor.



Map: /net/morticia/export/array1/docs/docs/tour/tour84.map

Frame:
0 Composition
1 MapFrame__modeler_output.i

Layer:
1 Annotation

Name /net/morticia/export/array1/docs/dc

6. In the Map Path Editor under **Frame**, click **MapFrame_modeler_output.img**.

   The type of **Layer** and **Layer Information** displays for the image. Note the path name for this image (located under **Layer Information** in the **Name** section).

7. Under **Frame**, click on **Composition**.

8. Under **Layer Information**, type the new file name or directory name in the **Name** text entry field.

9. Click **Apply** in the Map Path Editor.

   The changes you made are applied to the map composition.

   After you make each individual edit to each frame or layer, you must click **Apply**.

10. If you do not want the changes you have just made, click the **Reset** button.

11. When you are satisfied with your changes, save the file by selecting **File | Save** from the Map Path Editor menu bar.

For more information about cartography, see "Chapter 12: Cartography" *of the* ERDAS Field Guide.

CHAPTER FOUR

Vector Querying and Editing

Introduction

The IMAGINE Vector capabilities are designed to provide you with an integrated GIS package for raster and vector processing. The Vector tools in ERDAS IMAGINE are based on the ESRI data models, therefore ArcInfo vector coverages, ESRI shapefiles, and ESRI SDE vectors can be used in ERDAS IMAGINE with no conversion.

This tour guide explains how to edit vector layers

The data used in this tour guide are in the <IMAGINE_HOME>/examples directory. Replace <IMAGINE_HOME> with the directory where ERDAS IMAGINE is installed on your system (e.g., /usr/imagine/850).

A Digitizing Template is supplied in the information packet for ERDAS IMAGINE V8.6.

For more information about IMAGINE Vector, see "Chapter 21: IMAGINE Vector™".

This tour guide covers the following topics:

- creating new vector layers
- changing vector properties
- creating attributes

Approximate completion time for this tour guide is 55 minutes.

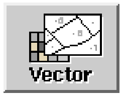
Query Vector Data

NOTE: The following section covers only the native Vector functionalities in ERDAS IMAGINE. If you have the IMAGINE Vector module, please see “Section V IMAGINE Vector™” *on page 625.*

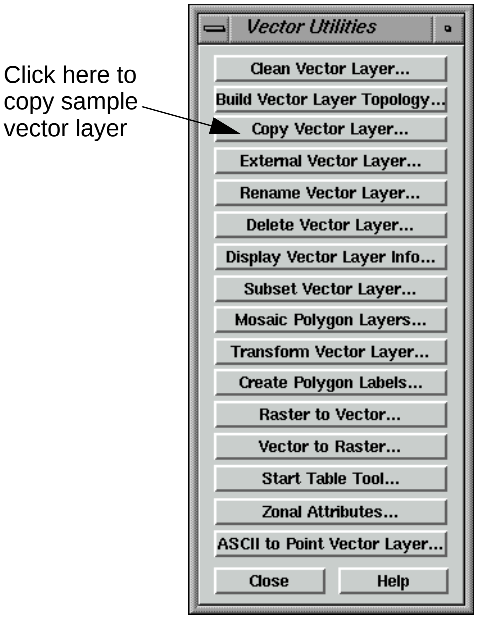
Copy Vector Data

Move to the directory where you want to create your workspace. Start ERDAS IMAGINE from this directory. Make sure this is a directory in which you have read/write permission.

1. Click on the Vector icon from the ERDAS IMAGINE icon panel.



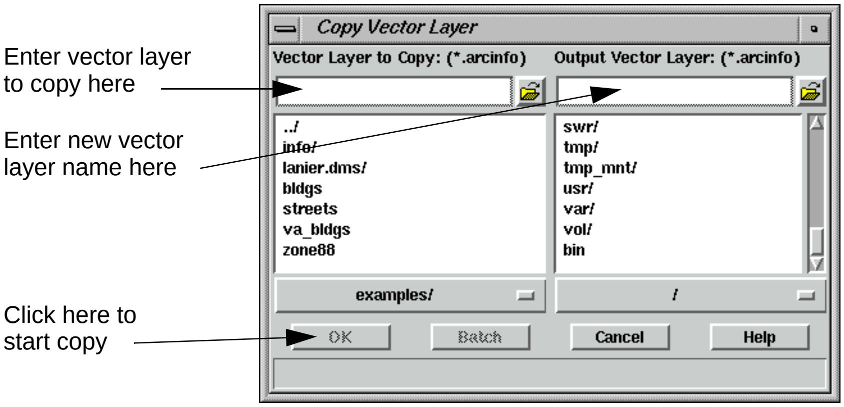
The **Vector Utilities** menu opens.


Click here to copy sample vector layer
Vector Utilities
Clean Vector Layer...
Build Vector Layer Topology...
Copy Vector Layer...
External Vector Layer...
Rename Vector Layer...
Delete Vector Layer...
Display Vector Layer Info...
Subset Vector Layer...
Mosaic Polygon Layers...
Transform Vector Layer...
Create Polygon Labels...
Raster to Vector...
Vector to Raster...
Start Table Tool...
Zonal Attributes...
ASCII to Point Vector Layer...
Close
Help

The vector utilities in this menu should NOT be run on open vector layers. Close the layer you are using before running the utility, and do NOT attempt to open the layer until the process is complete.

2. In the **Vector Utilities** menu, select **Copy Vector Layer**.

   The Copy Vector Layer dialog opens.


Enter vector layer to copy here
Enter new vector layer name here
Click here to start copy
Copy Vector Layer
Vector Layer to Copy: (*.arcinfo)
Output Vector Layer: (*.arcinfo)
../
info/
lanier.dms/
bldgs
streets
va_bldgs
zone88
swr/
tmp/
tmp_mnt/
usr/
var/
vol/
bin
examples/
/
OK
Batch
Cancel
Help

3. In the Copy Vector Layer dialog under **Vector Layer to Copy**, select the file named **zone88**.
4. Under **Output Vector Layer**, enter **zone88** in the directory of your choice. This should be the directory from which you started ERDAS IMAGINE.

5. Click **OK** in the Copy Vector Layer dialog.

   A Job Status dialog displays to track the progress of the function. When the Copy ArcInfo coverage process is complete, the files are copied and you are ready to proceed with this tour guide.

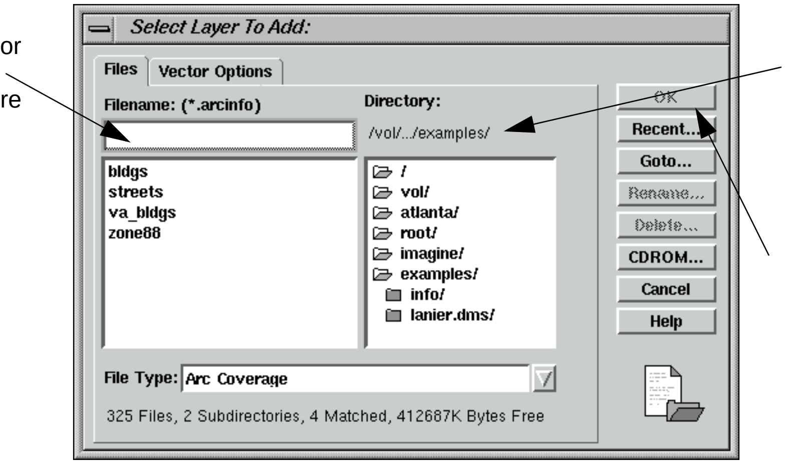
6. Click **Close** in the **Vector Utilities** menu to dismiss it.

Display Vector Layers

ERDAS IMAGINE should be running and a Viewer should be open. You must have completed the section, “Copy Vector Data” on page 95.

1. In the Viewer menu bar select **File | Open | Vector Layer**.

   The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Vector Options
Filename: (*.arcinfo)
bldgs
streets
va_bldgs
zone88
/
vol/
atlanta/
root/
imagine/
examples/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Directory: /vol/.../examples/

File Type: Arc Coverage

325 Files, 2 Subdirectories, 4 Matched, 412687K Bytes Free

Enter vector layer to display here

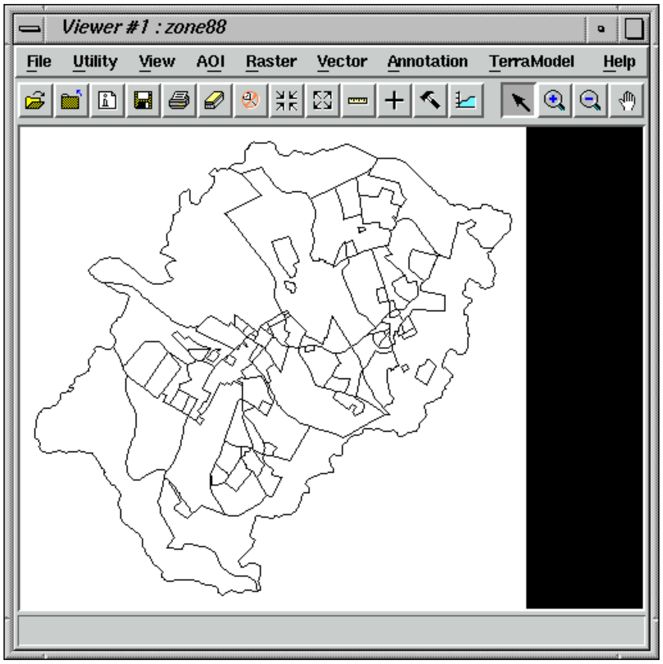
Confirm that the directory listed here is the directory in which you copied zone88

Click here to display the vector layer

2. Under **Filename** select **zone88** from the directory in which you saved it in the last section.

3. Click **OK** to display the layer in the Viewer.

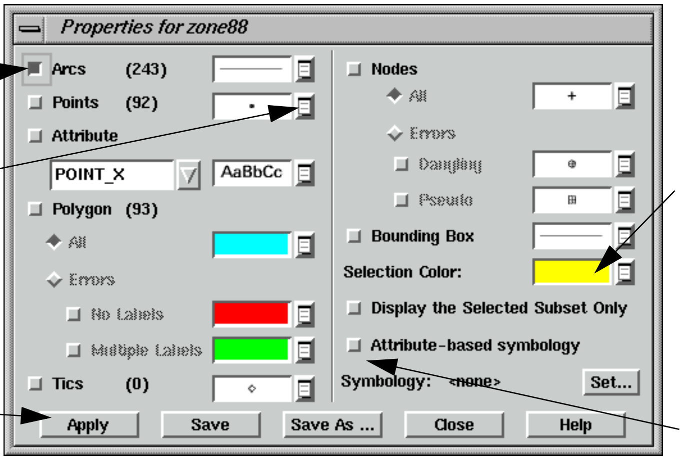
   The **zone88** polygon layer is displayed in the Viewer, similar to the following example:


Viewer #1 : zone88
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Change Vector Properties

1. Select **Vector | Viewing Properties** from the Viewer menu bar.

   The Properties dialog opens.


Properties for zone88
Arcs (243)
Points (92)
Attribute
POINT_X
AaBbCc
Polygon (93)
All
Errors
No Labels
Multiple Labels
Tics (0)
Nodes
All
Errors
Dangling
Pseudo
Bounding Box
Selection Color:
Display the Selected Subset Only
Attribute-based symbology
Symbology: <none>
Set...
Apply
Save
Save As ...
Close
Help

Click the check boxes to select the features to display. Use the popup lists to modify properties.

Right-hold this color patch to select another highlight color for selected features.

Click to apply new styles to the displayed vectors.

Click to use attribute-based symbology.

This dialog allows you to determine how and which vector features (lines, points, attributes, polygons, tics, and nodes) are displayed. You can also select the color to use for selected features.

In this example, the lines are currently displayed in the Viewer using the default styles shown in the Properties dialog.

2. In the Properties dialog, click to turn off the **Arcs** checkbox, then click to turn on **Points**.

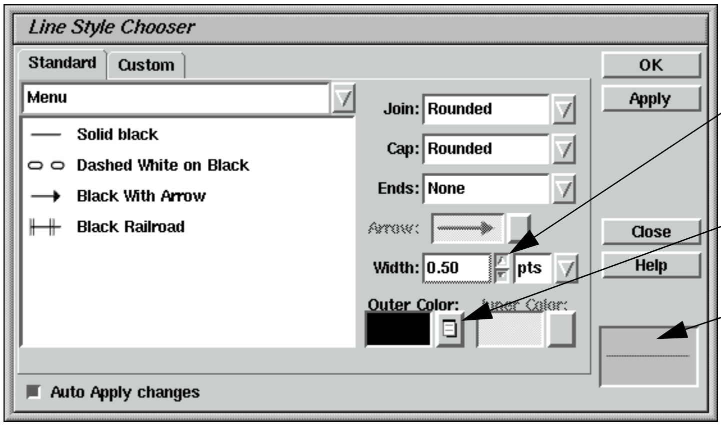
3. Click **Apply** in the Properties dialog.

   Now, points are displayed in the Viewer and lines are not. These are the polygon label points.

4. In the Properties dialog, click **Points**, **Arcs**, **Polygon**, and then **Apply**, so that lines and polygons are displayed and points are not.

5. Left-hold (for UNIX) or left-click (for PC) on the popup list next to **Arcs** and select **Other** to change the line style used.

   The Line Style Chooser dialog opens.



Line Style Chooser

Join: Rounded

Cap: Rounded

Ends: None

Width: 0.50 pts

Outer Color:

Inner Color:

Change line width here

Change line color here

Current line style is shown here

6. In the Line Style Chooser dialog, change the **Width** to **2.00** points.

7. Left-hold (UNIX) or left-click (PC) on the **Outer Color** popup list and select **Red**.

8. Click **Apply** and then **Close** in the Line Style Chooser dialog.

   The Line Style Chooser dialog closes. The new line style is reflected in the Properties dialog.

9. Click **Apply** in the Properties dialog to change the displayed vectors in the Viewer.

   The vectors are redrawn in the Viewer as thick, red lines.

10. In the Properties dialog, right-hold on the popup list next to **Arcs** and select **Other** again. The Line Style Chooser dialog opens.

11. Change the **Outer Color** back to **Black** and the **Width** back to **0.500** points, then click **Apply**.

12. Click **Close** in the Line Style Chooser dialog and then click **Apply** in the Properties dialog. The lines are redisplayed in black.

Display Attributes in the Viewer

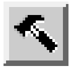
1. In the Properties dialog, click the **Points** and **Attribute** checkboxes, then click on the popup list under **Attribute** and select **ZONING**.
2. Click **Apply** in the Properties dialog.

   The polygon label points and zone numbers display in the Viewer.
3. In the Properties dialog, click the **Attribute** checkbox to deselect it, and then click **Apply**.
4. Click **Close** in the Properties dialog.
5. An Attention dialog displays, asking if you want to save these styles in a symbology file. Click **No**.

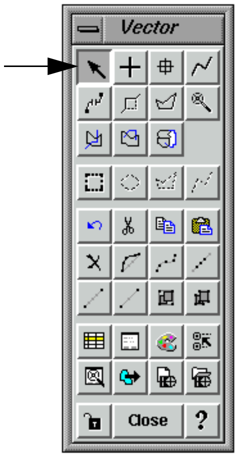
   The Properties dialog closes.

View Attributes

1. In the Viewer tool bar, click the Tools icon (or select **Vector | Tools** from the Viewer menu bar).



   The Vector tool palette displays



Use the Select tool to select features

NOTE: Depending on the package you have, your tool palette may include more icons than the one pictured above. If you have the IMAGINE Vector module, please see “Section V IMAGINE Vector™” *on page 625 for a description of the entire Vector tool palette.*

2. With your cursor in the Viewer, click on a polygon to select it (the Select tool is enabled by default in the Vector tool palette).

The selected polygon is highlighted in yellow.

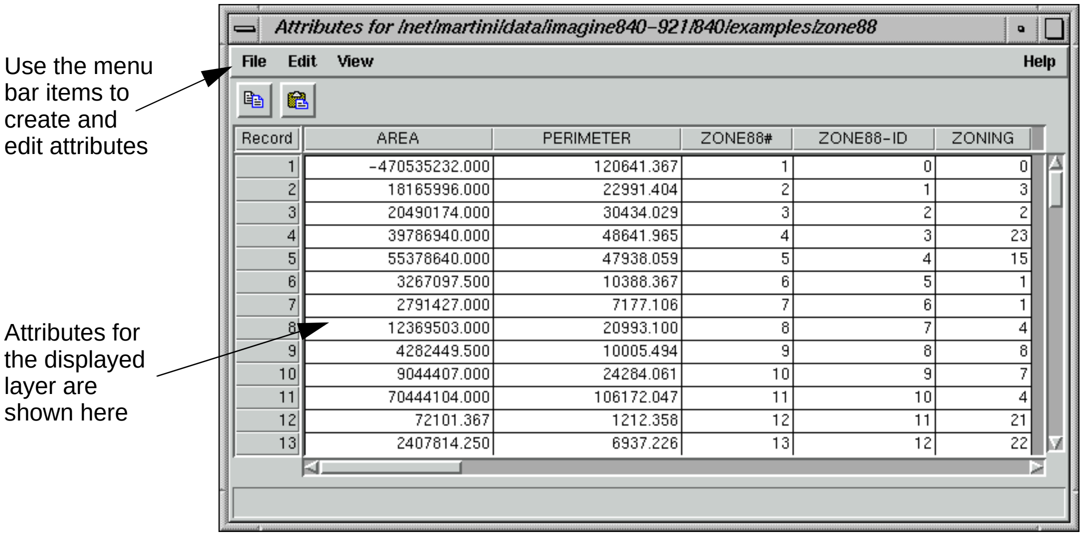
3. Shift-click on another polygon to add to the selection.

   Now two polygons are highlighted in yellow.

4. Click outside of the polygons (within the Viewer) to deselect everything.

5. In the Viewer menu bar, select **Vector** | **Attributes**.

   The Attributes dialog displays, as in the following example:


Attributes for /net/martini/data/imagine840-921/840/examples/zone88
File
Edit
View
Help

Use the menu bar items to create and edit attributes

Attributes for the displayed layer are shown here

Record	AREA	PERIMETER	ZONE88#	ZONE88-ID	ZONING
1	-470535232.000	120641.367	1	0	0
2	18165996.000	22991.404	2	1	3
3	20490174.000	30434.029	3	2	2
4	39786940.000	48641.965	4	3	23
5	55378640.000	47938.059	5	4	15
6	3267097.500	10388.367	6	5	1
7	2791427.000	7177.106	7	6	1
8	12369503.000	20993.100	8	7	4
9	4282449.500	10005.494	9	8	8
10	9044407.000	24284.061	10	9	7
11	70444104.000	106172.047	11	10	4
12	72101.367	1212.358	12	11	21
13	2407814.250	6937.226	13	12	22

   Polygon attributes are displayed in a CellArray. Therefore, you have access to the same tools that you use in other CellArrays.

6. In the Viewer, click on another polygon to select it.

   The polygon is highlighted in yellow in the Viewer, and the corresponding record in the Attributes CellArray is also highlighted.

7. Click on a record number under **Record** in the Attributes CellArray to select it.

   That record is highlighted in the CellArray and the corresponding polygon is highlighted in the Viewer.

8. With your cursor in the **Record** column of the Attributes CellArray, right-hold **Row Selection | Select All**.

   All features in both the CellArray and the Viewer are selected.

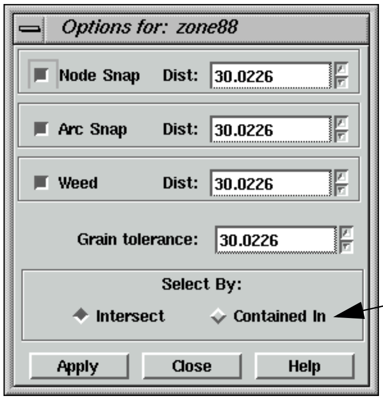
9. With your cursor in the **Record** column of the Attributes CellArray, right-hold **Row Selection | Select None** to deselect all features.

Use the Marquee Tools to Select Features

The marquee tool allows you to select a group of objects by dragging a box to contain them.

1. In the Viewer menu bar, select **Vector** | **Options**.

   The Options dialog opens.



Options for: zone88

☐ Node Snap Dist: 30.0226

☐ Arc Snap Dist: 30.0226

☐ Weed Dist: 30.0226

Grain tolerance: 30.0226

Select By:

● Intersect ○ Contained In

Click here to restrict selection to only those features within the marquee boundary

2. In the Options dialog under **Select By**, click the **Contained In** checkbox.
3. Click **Apply** in the Options dialog.

   When you use the marquee tools in the Vector tool palette now, only the features completely contained within the boundary of the marquee are selected.

4. Click the Marquee icon in the Vector tool palette.
5. Drag to draw a rectangle in the Viewer.

   When you release the mouse button, the polygons and lines contained within the rectangle are selected (outlined in yellow). The corresponding attributes are selected in the Attribute CellArray.

6. In the Viewer, click outside of the polygons to deselect all features.
7. Click the Line Selection icon in the Vector tool palette.
8. In the Viewer, click to start drawing a line over the vectors and either double-click or middle-click once (depending on how your Preferences are set) to terminate the line.

The features that intersect the line are selected in the Viewer and in the Attribute CellArray.

9. In the Viewer, click outside of the polygons to deselect everything.

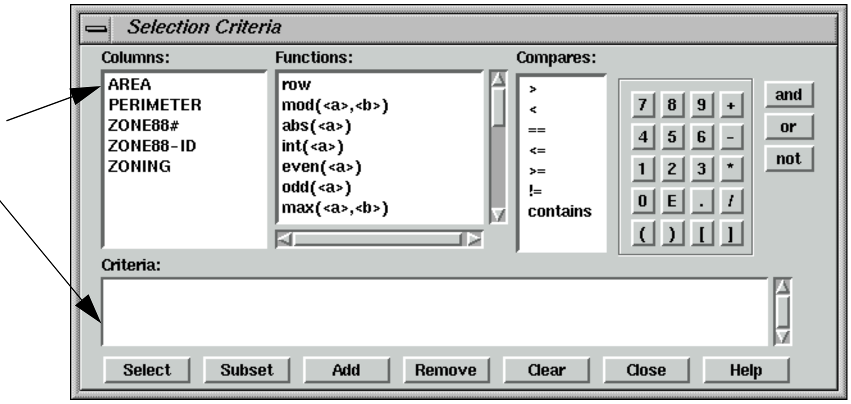
10. Click **Close** in the Options dialog.

Use the Criteria Function

This function allows you to specify the criteria upon which the automatic selection of CellArray rows is based.

1. With your cursor in the **Record** column of the Attributes CellArray, right-hold **Row Selection | Criteria**.

The Selection Criteria dialog opens.


Selection Criteria
Columns:
AREA
PERIMETER
ZONE88#
ZONE88-ID
ZONING
Functions:
row
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
max(<a>,<b>)
Compares:
>
<
==
<=
>=
!=
contains
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
and
or
not
Criteria:
Select
Subset
Add
Remove
Clear
Close
Help

Select items here to create criteria statement here

Next, you create a criteria statement to select only those polygons that have an area greater than 5,000,000 square feet.

2. In the Selection Criteria dialog, click on **AREA** under **Columns**.

$"AREA" is now written in the Criteria statement box at the bottom of the Selection Criteria dialog.

3. Under **Compares**, click on **>**.

The greater than symbol displays in the Criteria statement.

4. Use the numeric keypad to enter the number **5000000**.

The Criteria statement now reads:

```
$"AREA" > 5000000
```

5. Click **Select** to compare the attributes in the Attributes CellArray against this criteria statement.

Only those records that meet the criteria are selected and highlighted in the Viewer and Attributes CellArray.

Now, you further refine the criteria by limiting the selection to only those polygons that are both greater than 5,000,000 square feet and in zone 4.

6. In the Selection Criteria dialog, click the **and** button.
7. Under **Columns**, click **ZONING**.
8. Under **Compares**, click **==**.
9. In the numeric keypad, click **4**.

   The Criteria statement now reads:
   $"AREA" > 5000000 and $"ZONING" == 4

10. Click **Select** in the Selection Criteria dialog.

    All polygons greater the 5,000,000 square feet and in zone 4 are selected in the Viewer and in the Attribute CellArray.

11. Click **Close** in the Selection Criteria dialog.
12. In the Viewer, click outside of the polygons (within the Viewer) to deselect everything.
13. Click **Close** in the Vector tool palette.
14. Select **File | Close** from the Attributes dialog.
15. Select **File | Clear** from the Viewer.

Edit Vector Layers

For this section, display a Landsat TM raster layer, then overlay the vector layer you have been using.

ERDAS IMAGINE should be running and you should have a Viewer open.

1. Select **File | Open | Raster Layer** from the Viewer menu bar or click the Open icon on the tool bar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, select **germtm.img** from the file list.
3. Click the **Raster Options** tab at the top of the dialog, and then click on the **Fit to Frame** option so that the entire layer is visible in the Viewer.
4. Click **OK** in the Select Layer To Add dialog.

   The file **germtm.img** is displayed in the Viewer.

5. In the Viewer menu bar, select **File | Open | Vector Layer**.

The Select Layer To Add dialog opens.

6. In the Select Layer To Add dialog under **Filename**, select **zone88**.

   NOTE: If **zone88** *does not appear in the file list, click on the popup list next to* **File Type** *and select* **Arc Coverage**.

7. Click the **Vector Options** tab at the top of the dialog, and then confirm that the **Use Symbology** checkbox is turned off. Also, make sure that the **Clear Display** checkbox is turned off, so that the raster layer remains in the Viewer.

8. Click **OK** in the Select Layer To Add dialog.

   The vector layer is displayed over the raster layer. Since the vectors are black, they may not be easily visible.

9. With your cursor in the Viewer, right-hold **Quick View | Zoom | Rotate and Magnify Area**.

   A white rotation box displays in the Viewer and a Rotate/Magnify Instructions box also displays.

10. Drag the white rotation box so that it is positioned over the vector layer in the Viewer. Make it just big enough to cover the vectors, then double-click within the rotation box to magnify that area.

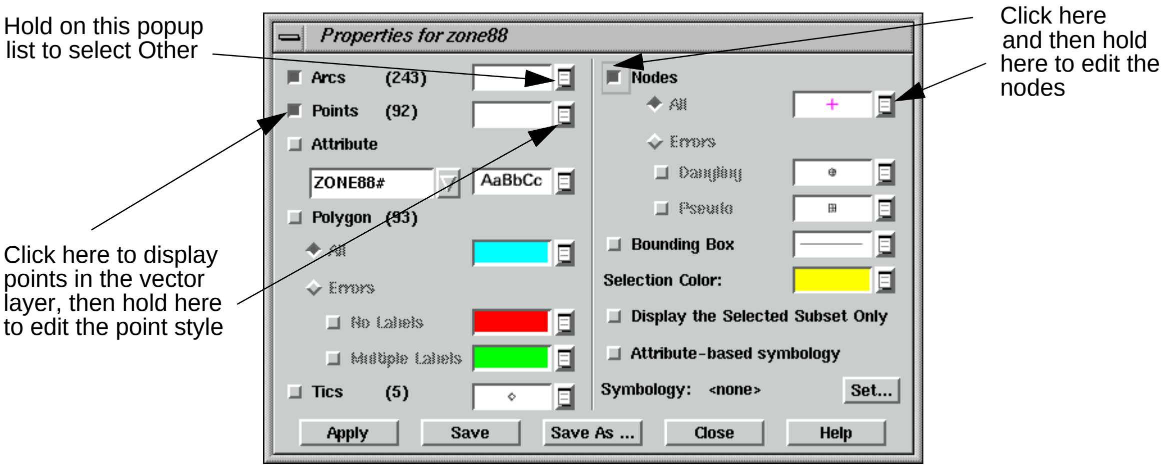
    The Rotate/Magnify Instructions box dismisses and both the raster and vector layers are magnified in the Viewer.

For other methods of zooming into an area of interest, see "Animated Zoom", "Box Zoom", *and* "Real-time Zoom", *starting on page 18.*

Change Viewing Properties

1. In the Viewer menu bar, select **Vector | Viewing Properties**.

   The Properties dialog opens.


Properties for zone88
Arcs (243)
Points (92)
Attribute
ZONE88#
AaBbCc
Polygon (93)
All
Errors
No Labels
Multiple Labels
Tics (5)
Nodes
All
Errors
Dangling
Pseudo
Bounding Box
Selection Color:
Display the Selected Subset Only
Attribute-based symbology
Symbology: <none>
Set...
Apply
Save
Save As ...
Close
Help

Hold on this popup list to select Other

Click here and then hold here to edit the nodes

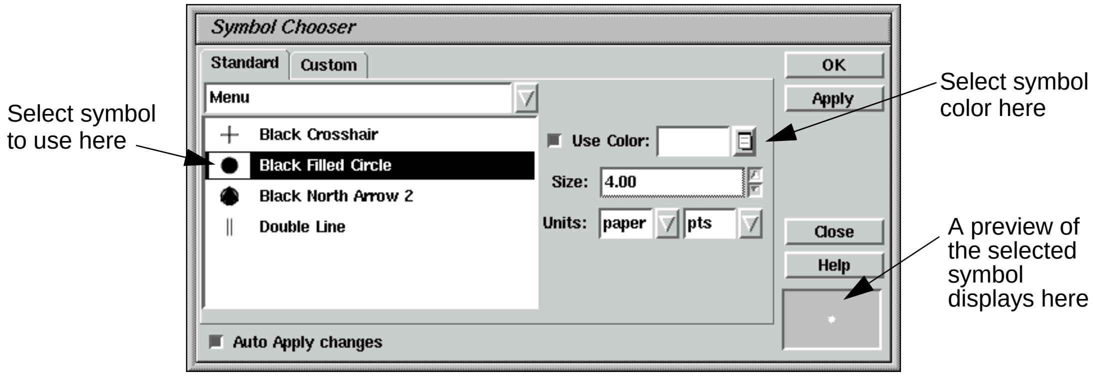
Click here to display points in the vector layer, then hold here to edit the point style

2. In the Properties dialog, hold on the popup list next to **Arcs** and select **Other**.

   The Line Style Chooser dialog opens.

3. In the Line Style Chooser dialog, change the **Width** to **2.00** points, and hold on the **Outer Color** popup list to select **White**.
4. Click **Apply** and then **Close** in the Line Style Chooser dialog.
5. Click **Points** in the Properties dialog to display points.
6. Hold on the popup list next to **Points** and select **Other** to change the style of the points so that they are more visible.

   The Symbol Chooser dialog opens.


Symbol Chooser
Standard
Custom
Menu
Black Crosshair
Black Filled Circle
Black North Arrow 2
Double Line
Use Color:
Size: 4.00
Units: paper pts
OK
Apply
Close
Help
Auto Apply changes

Select symbol to use here

Select symbol color here

A preview of the selected symbol displays here

7. In the Symbol Chooser dialog under **Menu**, click **Black Filled Circle**.
8. Hold on the **Use Color** popup list and select **White**.
9. Change the **Size** to **4.00** points.

10. Click **Apply** and then **Close** in the Symbol Chooser dialog.

    The new style is reflected in the Properties dialog (although you cannot see it because it is a white symbol against a white background).

11. In the Properties dialog, click **Nodes**, then hold on the popup list next to **All** and select **Other** to change the style of nodes.

    The Symbol Chooser dialog opens.

12. Change the **Size** of the symbol to **6.00** points.

13. Hold on the **Use Color** popup list and change the color of the symbol to something that is visible over **germtm.img**, such as **Magenta**.

14. When you have selected a color, click **Apply** and then **Close** in the Symbol Chooser dialog.

15. Click **Apply** in the Properties dialog.

    The vectors and points are drawn in white, and the nodes appear in the color you selected in step 13.

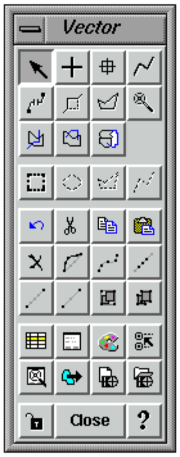
16. Click **Close** in the Properties dialog.

17. An Attention dialog displays, asking if you want to save your changes to a symbology file. Click **No**.

Use Editing Tools and Commands

1. In the Viewer menu bar, select **Vector | Tools.**

   The Vector tool palette displays.



2. In the Vector tool palette, move the selector over each of the icons to view the single-line help that describes them in the lower portion of the Viewer.

3. In the Viewer menu bar, select **Vector | Enable Editing**.

4. In the Viewer, click on a line that you want to edit.

   The selected line is highlighted in yellow and it is enclosed in a rotation box. You can simply hold and drag the entire box to move the line. You can hold and drag any of the handles on the box to enlarge or reduce the size of the line, or you can rotate the box.

5. In the Vector tool palette, click on the Split tool.

   Your pointer is now a white crosshair when you move it back into the Viewer.

6. Click somewhere on the selected line to split it into two lines.

   A node is placed where you clicked.

7. Click again on the line you split.

   Only that part of the line is selected, since it is now two lines.

8. Shift-click on the other part of the split line.

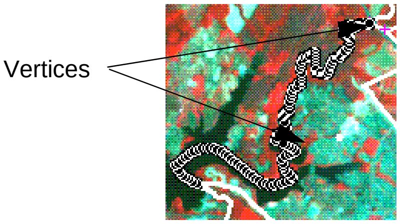
   Now both parts of the line are selected.

9. In the Viewer menu bar, select **Vector | Join**.

   The node is removed, and the two lines form a single line again.

10. With the line still selected, select **Vector | Reshape** from the Viewer menu bar.

    The vertices of the line display, similar to the following example.



    Each vertex is marked by a black dot in the example above.

11. Drag one of the vertices to a new location to reshape the line.

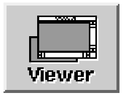
12. Select **Vector | Undo** from the Viewer menu bar to undo this edit.

    The line is restored to its original shape and it is deselected.

    NOTE: To delete a vertex, you can Shift-middle click. To add a vertex, middle-click.

Create New Vector Layer

1. Click the **Viewer** icon in the ERDAS IMAGINE icon panel.



A new Viewer opens.

2. In the new Viewer (Viewer #2) select **File | Open | Raster Layer**.

The Select Layer To Add dialog opens.

3. In the Select Layer To Add dialog, select **IMAGINE Image** from the **File Type** popup list.
4. Under **Filename**, click on the file **germtm.img**.
5. Click on the **Raster Options** tab at the top of the dialog and confirm that the **Fit to Frame** option is enabled.
6. Click **OK** in the Select Layer To Add dialog.

The file **germtm.img** displays in the Viewer.

7. In the Viewer #2 menu bar, select **File | New | Vector Layer**.

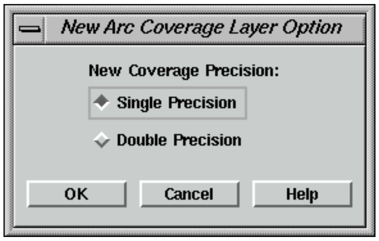
The New Vector Layer dialog opens.

8. In the New Vector Layer dialog under **Vector Layer**, enter a name for the new layer, such as **zone88subset**, in the directory of your choice.

Vector coverages include Arc Coverage, SDE Vector Layer, and Shapefile.

9. Click **OK** in the New Vector Layer dialog.

The New Arc Coverage Layer Option dialog opens.


New Arc Coverage Layer Option

New Coverage Precision:

● Single Precision

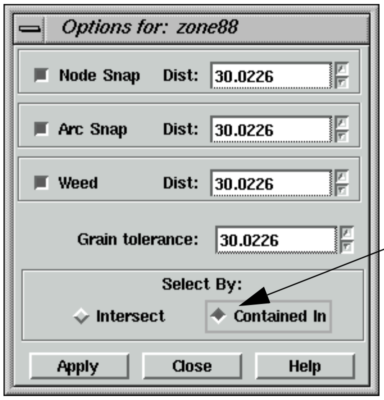
○ Double Precision

10. Confirm that **Single Precision** is selected.
11. Click **OK** in the New Arc Coverage Layer Option dialog.

The title bar in Viewer #2 reflects the name of the new vector layer you are creating.

12. In the menu bar of Viewer #1, select **Vector | Options**.

    The Options dialog opens.



Options for: zoone88

☐ Node Snap Dist: 30.0226

☐ Arc Snap Dist: 30.0226

☐ Weed Dist: 30.0226

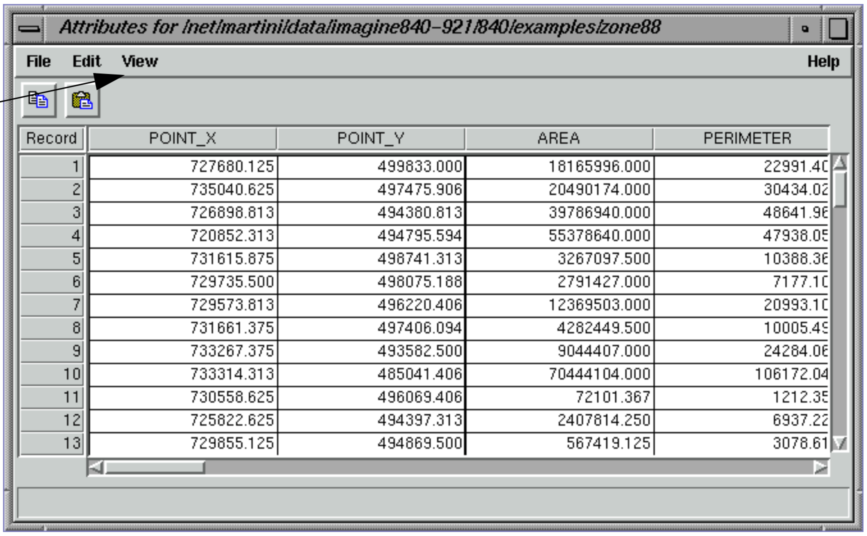
Grain tolerance: 30.0226

Select By: ○ Intersect ● Contained In

Click here to select Contained In

13. In the Options dialog under **Select By**, click **Contained In**.
14. Click **Apply** and then **Close** in the Options dialog.
15. In the menu bar of Viewer #1, select **Vector | Attributes**.

    The Attributes dialog opens.



Attributes for /net/martini/data/imagine840-921/840/examples/zone88

Record	POINT_X	POINT_Y	AREA	PERIMETER
1	727680.125	499833.000	18165996.000	22991.4
2	735040.625	497475.906	20490174.000	30434.0
3	726898.813	494380.813	39786940.000	48641.9
4	720852.313	494795.594	55378640.000	47938.0
5	731615.875	498741.313	3267097.500	10388.3
6	729735.500	498075.188	2791427.000	7177.1
7	729573.813	496220.406	12369503.000	20993.1
8	731661.375	497406.094	4282449.500	10005.4
9	733267.375	493582.500	9044407.000	24284.0
10	733314.313	485041.406	70444104.000	106172.04
11	730558.625	496069.406	72101.367	1212.3
12	725822.625	494397.313	2407814.250	6937.2
13	729855.125	494869.500	567419.125	3078.61

Click here to be sure that Point Attributes is selected

16. In the Attributes dialog menu bar, click on **View** and make sure that **Point Attributes** is selected.

17. In Viewer #1, click on the Zoom Out by 2 icon.

Export Zoning Attributes

1. Click on the Ellipse Marquee icon in the Vector tool palette.

2. Shift-drag to draw a circle over the vector layer in Viewer #1.

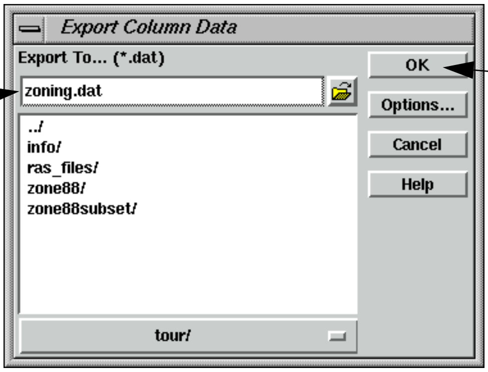
The lines and points completely contained within the circle are selected in the Viewer and in the Attributes CellArray. You are going to export the **ZONING** attributes for the selected rows and import them into the new vector layer you are creating in Viewer #2.

3. Click the **ZONING** column header in the Attributes CellArray to select that column (use the bottom scroll bar to view the **ZONING** column).

The **ZONING** column is highlighted in green.

4. With your cursor in a column header of the Attributes CellArray, right-hold **Column Options | Export**.

The Export Column Data dialog opens.



Export Column Data

Export To... (*.dat)

zoning.dat

../
info/
ras_files/
zone88/
zone88subset/

tour/

Enter a name for the ASCII file here

Click here to export data to an ASCII file

5. In the Export Column Data dialog, enter a name for the ASCII file you are creating, such as **zoning**. The .dat extension is added automatically.

6. Click **OK** in the Export Column Data dialog.

7. Select **Vector | Copy** from the Viewer #1 menu bar.

The selected lines and points are copied into the paste buffer.

8. Click anywhere in Viewer #2, then select **Vector | Paste** from the Viewer #2 menu bar.

   The selected vectors are displayed in Viewer #2.

9. In Viewer #2, click outside of all lines and points of the vector layer to deselect everything.

10. In the Attributes dialog, select **File | Close**.

11. In the menu bar of Viewer #2, select **File | Save | Top Layer**.

12. In the menu bar of Viewer #2, select **Vector | Attributes**.

    The Attributes dialog displays, but it is empty because this new layer has no attribute data. It must be cleaned or built. You can also create attributes using the **Edit | Create Attributes** option of the Attributes dialog. Use this method now, then run Build later.

Create Attributes

1. In the Attributes dialog, select **View | Point Attributes** to select point attributes for display.

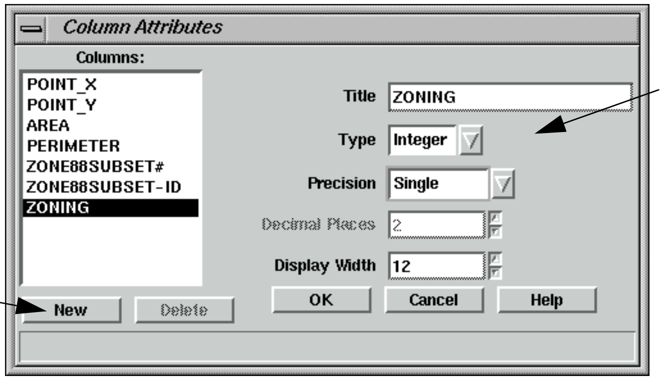
2. In the Attributes dialog, select **Edit | Create Attributes**.

   The Attributes CellArray fills with the basic point attributes.

   Next, you create a new column in the Attributes CellArray for the **ZONING** attribute that you exported from the original Attributes CellArray.

3. In the Attributes dialog, select **Edit | Column Attributes**.

   The Column Attributes dialog opens.



POINT_X
POINT_Y
AREA
PERIMETER
ZONE88SUBSET#
ZONE88SUBSET-ID
ZONING

Title: ZONING
Type: Integer
Precision: Single
Decimal Places: 2
Display Width: 12

Specify column parameters here

Click here to create a new column

4. In the Column Attributes dialog, click **New** to add a new column to the CellArray.

   The options on the right side of the dialog are now enabled, so that you can define the parameters of the new column.

5. For the column **Title**, enter **ZONING**. Press Return on your keyboard.

6. For **Type**, click on the popup list and select **Integer**.

7. For **Precision**, click on the popup list and select **Single**.

8. For **Display Width**, accept the default of **12**.

9. Click **OK** in the Column Attributes dialog to create the new column.

   The Attributes CellArray now has a new column called **ZONING**. This new column is placed to the right of the last column.

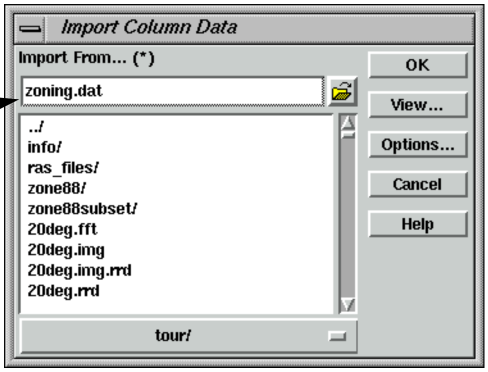
10. Click in the header of this new column (**ZONING**) to select it.

    The column is highlighted in blue.

11. With your cursor in the **Record** column, right-hold **Row Selection | Select All**.

12. With your cursor in a column header, right-hold **Column Options | Import**.

    The Import Column Data dialog opens.



Import Column Data
Import From... (*)
OK
View...
Options...
Cancel
Help
../
info/
ras_files/
zone88/
zone88subset/
20deg.fft
20deg.img
20deg.img.rrd
20deg.rrd
tour/

zoning.dat

Enter ASCII file name here

13. In the Import Column Data dialog, enter the name of the ASCII file that you created in the Export Column Data dialog in step 5. (i.e., **zoning.dat**).

14. Click **OK**.

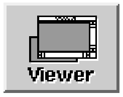
    The Attributes CellArray now has the same **ZONING** column and attributes as the original Attributes CellArray.

15. Select **File | Close** from the menu bar of Viewer #2. When asked if you would like to save changes, click **Yes**.

    The Attributes dialog automatically closes.

Create a Simple Shapefile Layer

1. Click the **Viewer** icon in the ERDAS IMAGINE icon panel.



A new Viewer opens.

2. In the new Viewer (Viewer #2) select **File | Open | Raster Layer**.

The Select Layer To Add dialog opens.

3. In the Select Layer To Add dialog, select **IMAGINE Image** from the **File Type** popup list.

4. Under **Filename**, click on the file **germtm.img**.

5. Click on the **Raster Options** tab at the top of the dialog and confirm that the **Fit to Frame** option is enabled.

6. Click **OK** in the Select Layer To Add dialog.

The file **germtm.img** displays in the Viewer.

7. With your cursor in the Viewer, right-hold **Quick View** | **Zoom** | **Rotate and Magnify Area**.

A white rotation box displays in the Viewer and a Rotate/Magnify Instructions box also displays.

8. Drag the white rotation box so that it is positioned over the same area that was covered by the zone88 vector coverage. Double-click within the rotation box to magnify that area.

The Rotate/Magnify Instructions box dismisses and the raster layer is magnified in the Viewer.

For other methods of zooming into an area of interest, see "Animated Zoom", "Box Zoom", *and* "Real-time Zoom", *starting on page 18.*

9. In the Viewer #2 menu bar, select **File | New | Vector Layer**.

The New Vector Layer dialog opens.

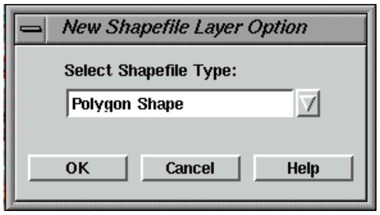
10. Under **File of Type**, select **Shapefile (*.shp)** from the pulldown list.

11. In the New Vector Layer dialog under **Vector Layer**, enter a name for the new layer, such as **zone88shapefile**, in the directory of your choice.

Vector coverages include Arc Coverage, SDE Vector Layer, and Shapefile.

12. Click **OK** in the New Vector Layer dialog.

The New Shapefile Layer Option dialog opens.


New Shapefile Layer Option
Select Shapefile Type:
Polygon Shape
OK
Cancel
Help

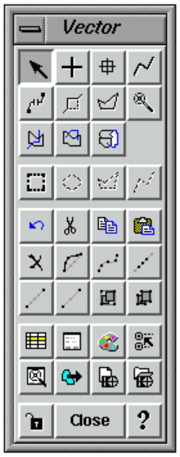
13. Select **Polygon Shape** from the pulldown list.

14. Click **OK** in the New Shapefile Layer Options dialog.

Create a Shapefile Coverage

1. In the Viewer tool bar, click the Tools icon (or select **Vector | Tools** from the Viewer menu bar).

The Vector tool palette displays

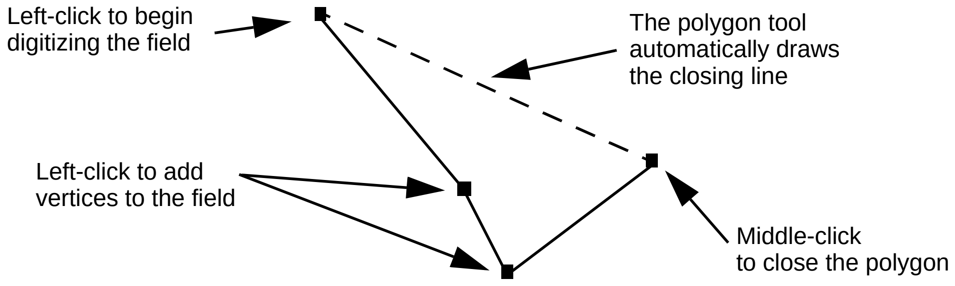

Vector
Close
?

2. In the Vector tool palette, click on the Polygon tool .

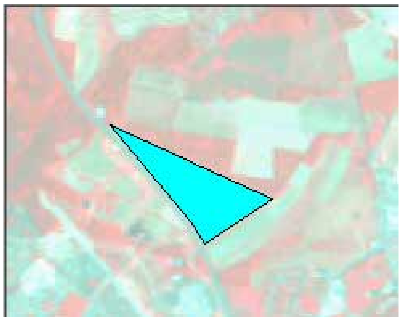
For more information on using the Shapefile editing tools, see the Vector Tools Diagram page in the Vector On-Line Manual.

3. Using the Inquire Cursor, find the triangular field at X: 726102.951800, Y: 497901.936911.

4. Left-click at the north end of the field and begin digitizing a shapefile coverage of the field.


Left-click to begin digitizing the field
The polygon tool automatically draws the closing line
Left-click to add vertices to the field
Middle-click to close the polygon

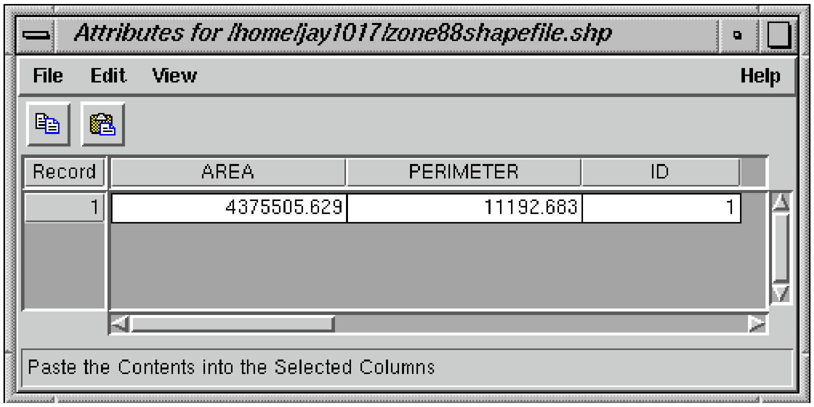
Your polygon should look something like this:



NOTE: The raster layer of these images have been washed out for the purposes of illustration.

5. Open the Vector Attributes table by selecting **Vector | Attributes** from the Viewer menu bar or by clicking on the **Vector Attributes** button in the Vector Tool palette.

6. The Vector Attributes table displays:


File
Edit
View
Help

Attributes for /home/jay1017/zone88shapefile.shp

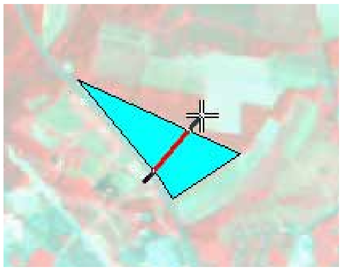
Record	AREA	PERIMETER	ID
1	4375505.629	11192.683	1

Paste the Contents into the Selected Columns

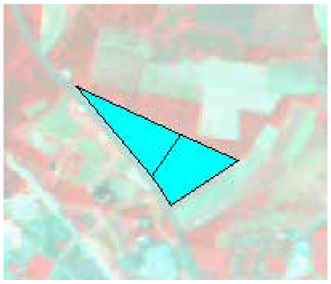
Editing the Shapefile Layer

7. To divide this polygon into two polygons, select the **Polygon Split** tool from the Vector Tools palette.

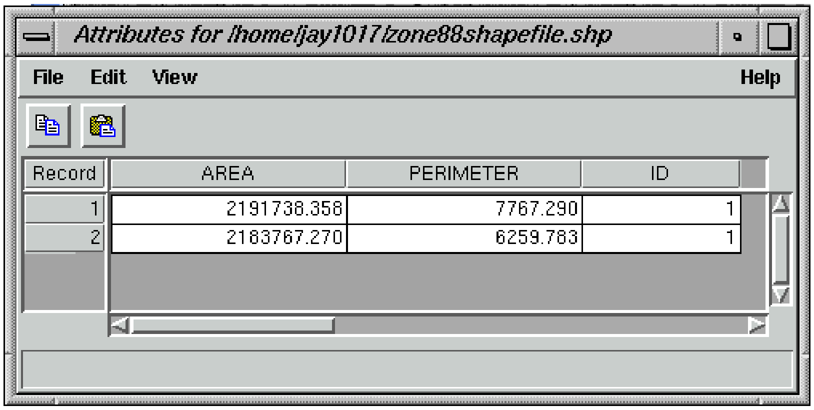
8. Left-click once just outside the left side of the polygon. Move your cursor across the middle and outside of the right side of the polygon so that the split line divide the polygon into no more than two parts. It should look similar to this:



9. Middle-click to end the split line. The polygon has split into two polygons and should look like this:



10. Notice that the Area and Perimeter columns were automatically updated in the Vector Attributes table:

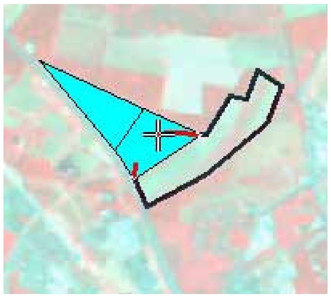


Attributes for /home/jay1017/zone88shapefile.shp

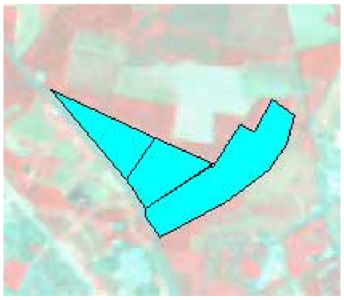
Record	AREA	PERIMETER	ID
1	2191738.358	7767.290	1
2	2183767.270	6259.783	1

11. Next you will create a new polygon that shares a common border with an existing polygon. To do this, select the Append Polygon tool from the Vector tools palette.

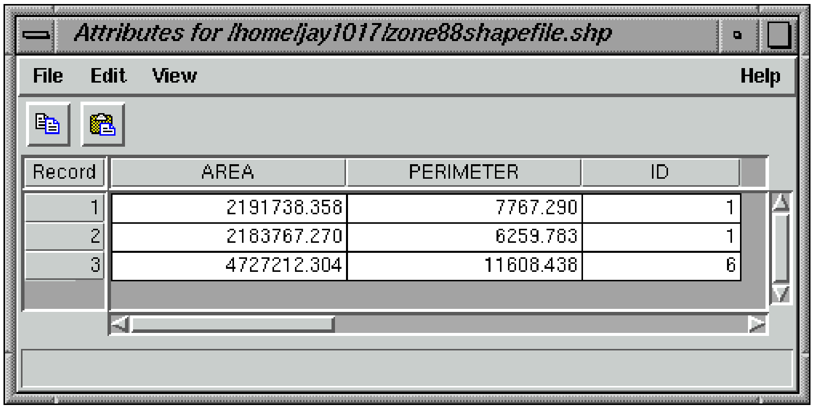
12. To begin digitizing the field below the original polygon, left-click *inside* of the existing polygon. Continue to left-click to add vertices until you have digitized the entire field. Your line should end up within the same polygon in which you started and look like this:



13. Double-click to close the polygon. The new polygon should look like this:



14. Again note that the columns in the Vector Attributes Table update to show the area and perimeter of the new polygon.

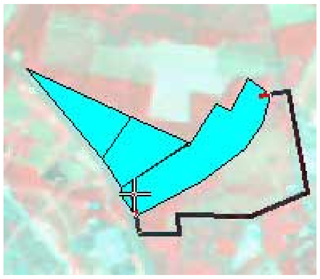


Attributes for /home/jay1017/zone88shapefile.shp

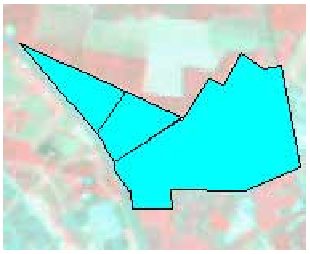
Record	AREA	PERIMETER	ID
1	2191738.358	7767.290	1
2	2183767.270	6259.783	1
3	4727212.304	11608.438	6

15. Next you will reshape this polygon so that it contains more of the field. To do this, you will need to select the Replace Polygon tool from the Vector tools palette.

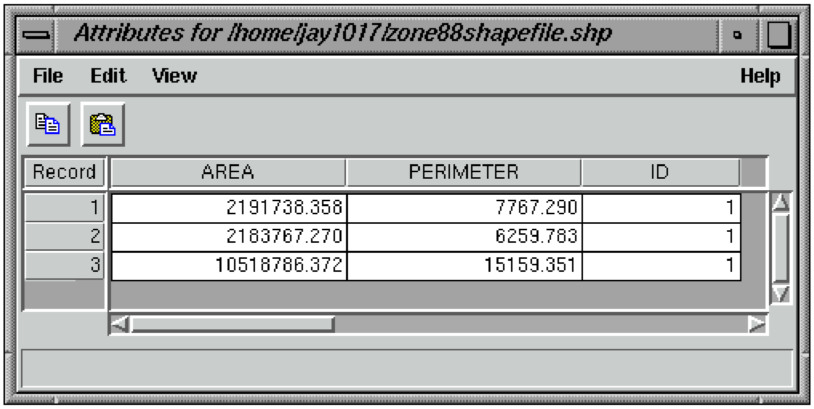
16. Again, start your digitizing by left-clicking inside of the polygon you want to reshape. Continue left-clicking to add vertices to the polygon. Your line should end up within the same polygon in which you started and look similar to this:



17. Double-click inside of the starting polygon to reshape the existing polygon to include the newly digitized area. Your new polygon should look like this:



18. The Area and Perimeter values in the Vector Attributes table reflect the dimensions of the polygon:



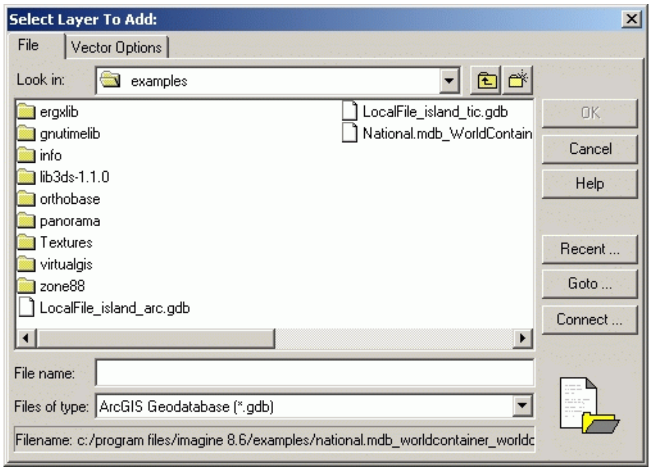
Record	AREA	PERIMETER	ID
1	2191738.358	7767.290	1
2	2183767.270	6259.783	1
3	10518786.372	15159.351	1

19. Click **File | Close** from the Viewer menu bar. When asked if you would like to save your changes, click **Yes**.

The Attributes dialog automatically closes.

Open a Personal Geodatabase

1. In a new viewer, click the **Open Layer** button and select ArcGIS Geodatabase (*.gdb) from the pull down menu in **Files of Type**.
2. Click on the **Connect** button that appears in the Select Layer To Add dialog to connect to the geodatabase.

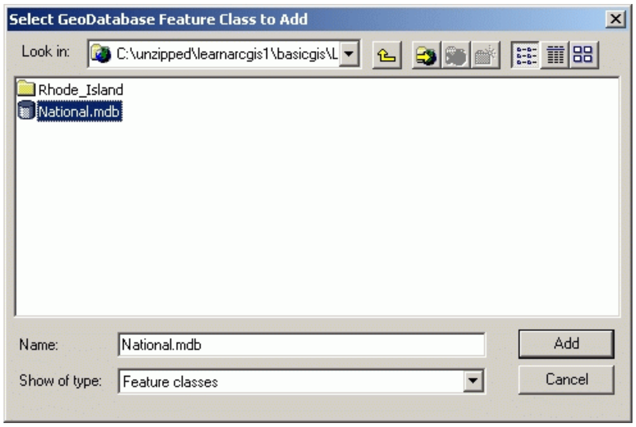

Select Layer To Add:
File
Vector Options
ergxlib
gnutimelib
info
lib3ds-1.1.0
orthobase
panorama
Textures
virtualgis
zone88
LocalFile_island_arc.gdb
LocalFile_island_tic.gdb
National.mdb_WorldContain
OK
Cancel
Help
Recent ...
Goto ...
Connect ...
File name:

Look in: examples

Files of type: ArcGIS Geodatabase (*.gdb)

Filename: c:/program files/imagine 8.6/examples/national.mdb_worldcontainer_worldc

The Select GeoDatabase Feature Class to Add dialog appears.

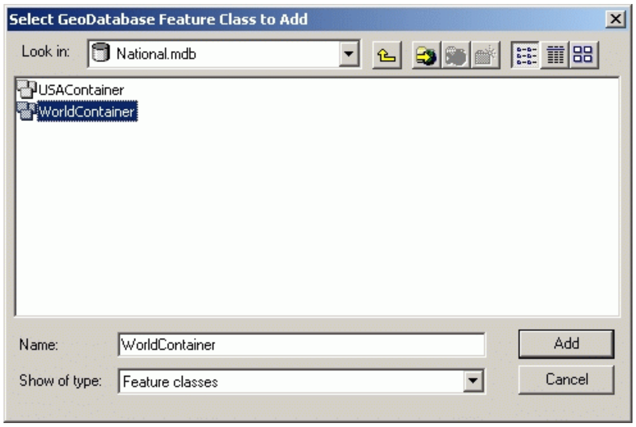

Select GeoDatabase Feature Class to Add
Rhode_Island
National.mdb
Add
Cancel

Look in: C:\unzipped\learnarcgis1\basicgis\L

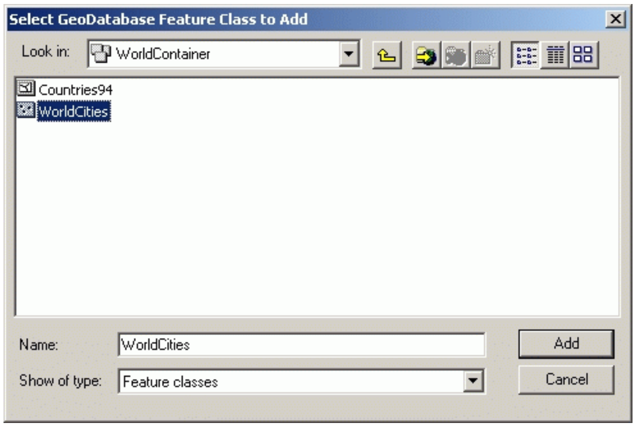
Name: National.mdb

Show of type: Feature classes

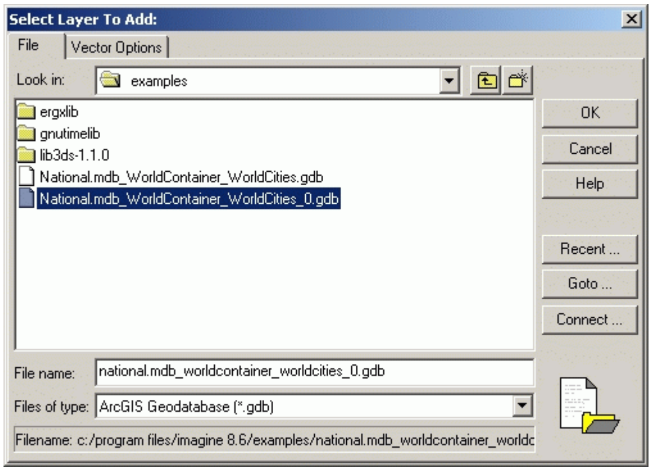
3. Select the directory where your personal geodatabase is stored. Select the .mdb file you wish to use and click Add.
4. The list of feature datasets appears in the dialog. Select the dataset you wish to use and click Add.


Select GeoDatabase Feature Class to Add
Look in:
National.mdb
USAContainer
WorldContainer
Name:
WorldContainer
Show of type:
Feature classes
Add
Cancel

5. Next, a list of feature classes will appear. Choose the one you want to add and click Add.


Select GeoDatabase Feature Class to Add
Look in:
WorldContainer
Countries94
WorldCities
Name:
WorldCities
Show of type:
Feature classes
Add
Cancel

6. This connection will be mapped for you to the proxy file, and you will be brought back to the Select Layer To Add dialog where it will be listed in a directory with any other mapped connections.


Select Layer To Add:
File
Vector Options
ergxlib
gnutimelib
lib3ds-1.1.0
National.mdb_WorldContainer_WorldCities.gdb
National.mdb_WorldContainer_WorldCities_0.gdb
OK
Cancel
Help
Recent ...
Goto ...
Connect ...

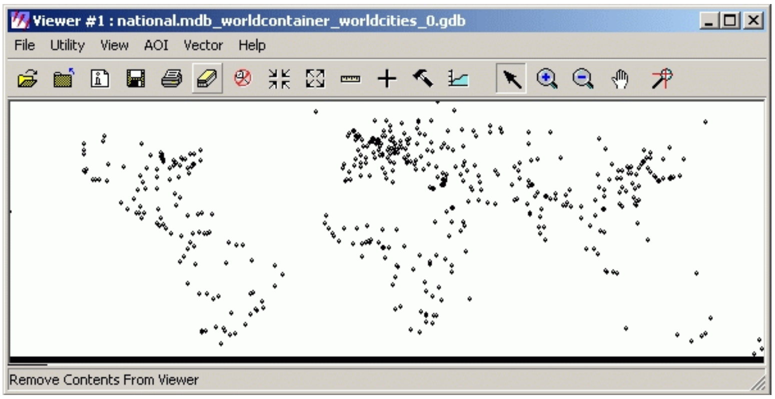
Look in: examples

File name: national.mdb_worldcontainer_worldcities_0.gdb

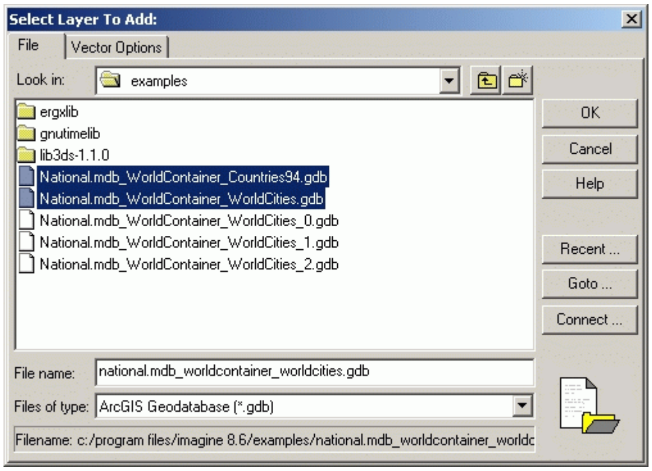
Files of type: ArcGIS Geodatabase (*.gdb)

Filename: c:/program files/imagine 8.6/examples/national.mdb_worldcontainer_worldc

7. Select the .gdb file you want to view. From this point on you can treat this like any other vector layer. You can access it and edit it just like any other vector data. You may also rename the file by right clicking on the file name and selecting rename. Click OK in the dialog to display the information in a viewer.


Viewer #1 : national.mdb_worldcontainer_worldcities_0.gdb
File
Utility
View
AOI
Vector
Help
Remove Contents From Viewer

You can also choose several mapped layers at once to display in a viewer by using your control key to select more than one feature class.

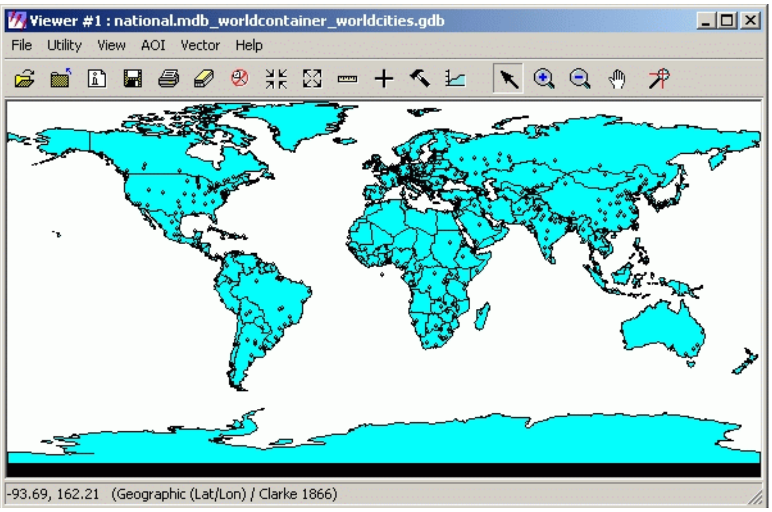

Select Layer To Add:
File
Vector Options
Look in:
examples
ergxlib
gnutimelib
lib3ds-1.1.0
National.mdb_WorldContainer_Countries94.gdb
National.mdb_WorldContainer_WorldCities.gdb
National.mdb_WorldContainer_WorldCities_0.gdb
National.mdb_WorldContainer_WorldCities_1.gdb
National.mdb_WorldContainer_WorldCities_2.gdb
OK
Cancel
Help
Recent ...
Goto ...
Connect ...
File name:
Files of type:

File name: national.mdb_worldcontainer_worldcities.gdb

Files of type: ArcGIS Geodatabase (*.gdb)

Filename: c:/program files/imagine 8.6/examples/national.mdb_worldcontainer_worldc

After selecting two or more layers and clicking OK, all selected layers display in the viewer:

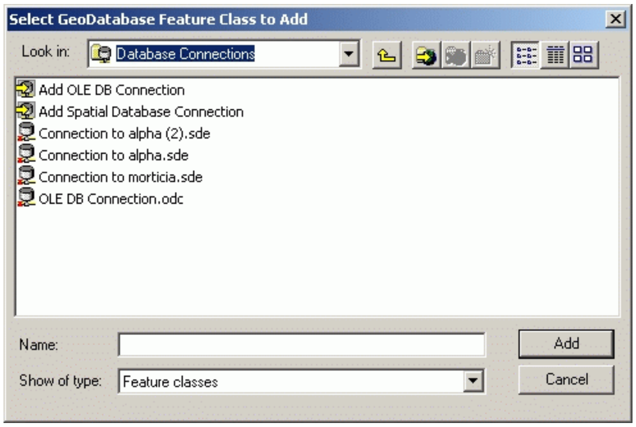

Viewer #1 : national.mdb_worldcontainer_worldcities.gdb
File
Utility
View
AOI
Vector
Help

-93.69, 162.21 (Geographic (Lat/Lon) / Clarke 1866)

Open An Enterprise Geodatabase

1. Click the Open Layer button in a new viewer. In the Files of Type pull down menu, choose ArcGIS Geodatabase .gdb as your file type.

2. Notice that this adds a **Connect** button to the dialog. Click **Connect**, and in the **Look in** pull down menu, select **Database Connections**.

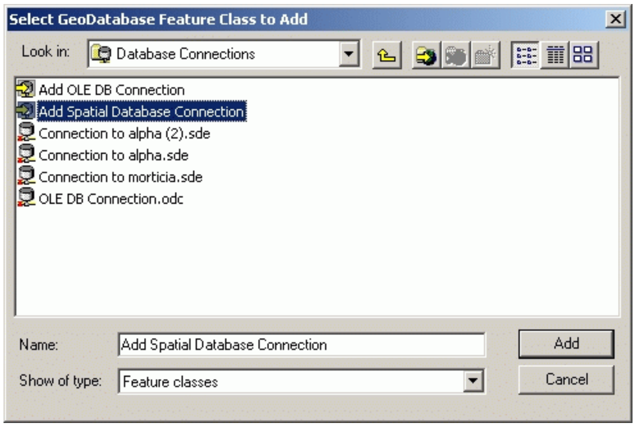

Select GeoDatabase Feature Class to Add
Add OLE DB Connection
Add Spatial Database Connection
Connection to alpha (2).sde
Connection to alpha.sde
Connection to morticia.sde
OLE DB Connection.odc
Add
Cancel

Look in: Database Connections

Name:

Show of type: Feature classes

3. Choose Add Spatial Database Connection and click Add.

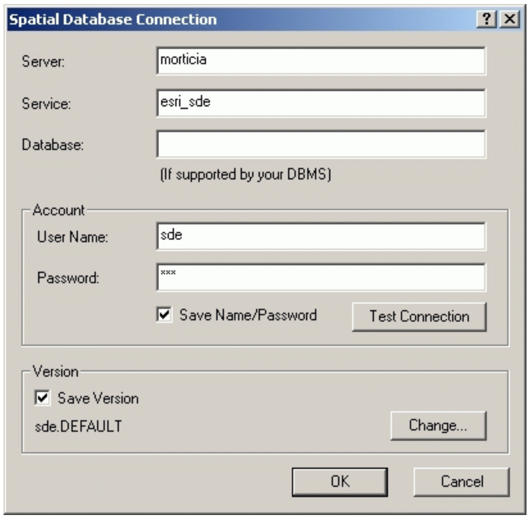

Select GeoDatabase Feature Class to Add
Add OLE DB Connection
Add Spatial Database Connection
Connection to alpha (2).sde
Connection to alpha.sde
Connection to morticia.sde
OLE DB Connection.odc
Add
Cancel

Look in: Database Connections

Name: Add Spatial Database Connection

Show of type: Feature classes

4. The Spatial Database Connection dialog appears.



Server: morticia

Service: esri_sde

Database:

(If supported by your DBMS)

Account

User Name: sde

Password: ***

☑ Save Name/Password

Version

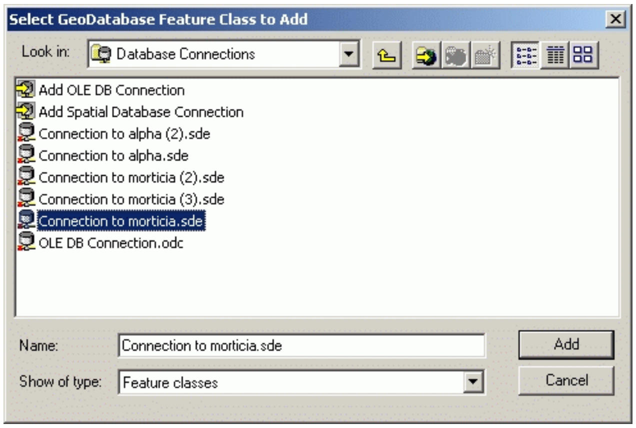
☑ Save Version

sde.DEFAULT

You should see your System Administrator to find out exactly what you should use as your server, service, database, and account information. The information used here is for instructional purposes only.

Click OK in the Spatial Database Connection dialog.

5. You are brought back to the Select Geodatabase Feature Class to Add dialog, and a connection to the server will be showing. Select this connection and click Add.



Look in: Database Connections

- Add OLE DB Connection
- Add Spatial Database Connection
- Connection to alpha (2).sde
- Connection to alpha.sde
- Connection to morticia (2).sde
- Connection to morticia (3).sde
- Connection to morticia.sde
- OLE DB Connection.odc

Name: Connection to morticia.sde

Show of type: Feature classes

You will be connected to the database where you can select a feature to create a proxy file that will serve as a connection to the server and feature classes. This proxy file will reside in the directory where your file chooser is located. If you wish to rename the file, right click and select rename. To access the feature classes in the database, simply double click the proxy file name.

CHAPTER FIVE

Classification

Introduction

Classification is the process of sorting pixels into a finite number of individual classes, or categories of data based on their data file values. If a pixel satisfies a certain set of criteria, then the pixel is assigned to the class that corresponds to that criteria.

In this tour guide, you perform a basic unsupervised classification of an image file (.img).

All of the data used in this tour guide are in the <IMAGINE_HOME>/examples directory. You should copy the **germtm.img** *file to a different directory so that you can have write permission to this file.*

For more detailed information on Classification techniques, please see "Advanced Classification" on page 455.

Approximate completion time for this tour guide is 20 minutes.

Use Unsupervised Classification

This section shows you how to create a thematic raster layer by letting the software identify statistical patterns in the data without using any ground truth data.

ISODATA Classifier

ERDAS IMAGINE uses the ISODATA algorithm to perform an unsupervised classification. The ISODATA clustering method uses the minimum spectral distance formula to form clusters. It begins with either arbitrary cluster means or means of an existing signature set. Each time the clustering repeats, the means of these clusters are shifted. The new cluster means are used for the next iteration.

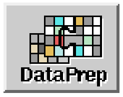
The ISODATA utility repeats the clustering of the image until either a maximum number of iterations has been performed, or a maximum percentage of unchanged pixel assignments has been reached between two iterations.

Performing an unsupervised classification is simpler than a supervised classification because the signatures are automatically generated by the ISODATA algorithm.

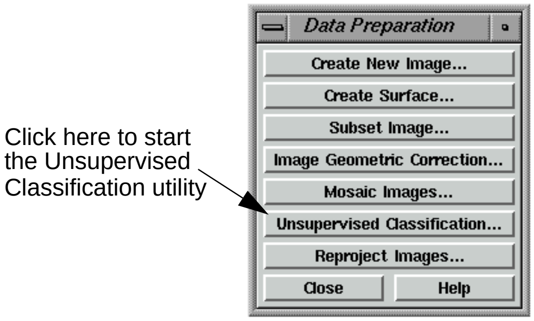
In this example, you generate a thematic raster layer using the ISODATA algorithm.

You must have ERDAS IMAGINE running.

1. Click on the DataPrep icon in the ERDAS IMAGINE icon panel.



The **Data Preparation** menu opens.

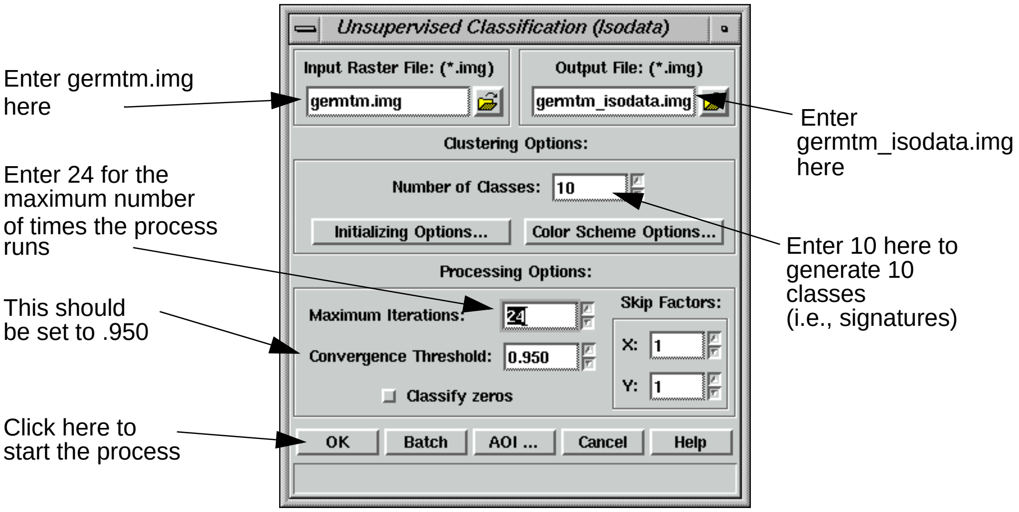

Data Preparation
Create New Image...
Create Surface...
Subset Image...
Image Geometric Correction...
Mosaic Images...
Unsupervised Classification...
Reproject Images...
Close
Help

Click here to start the Unsupervised Classification utility

Generate Thematic Raster Layer

1. Select **Unsupervised Classification** from the **Data Preparation** menu to perform an unsupervised classification using the ISODATA algorithm.

The Unsupervised Classification (Isodata) dialog opens.


Unsupervised Classification (Isodata)
Clustering Options:
Initializing Options...
Color Scheme Options...
Processing Options:
Skip Factors:
OK
Batch
AOI ...
Cancel
Help

Input Raster File: (*.img) germtm.img

Output File: (*.img) germtm_isodata.img

Number of Classes: 10

Maximum Iterations: 24

Convergence Threshold: 0.950

☐ Classify zeros

X: 1

Y: 1

Enter germtm.img here

Enter germtm_isodata.img here

Enter 24 for the maximum number of times the process runs

Enter 10 here to generate 10 classes (i.e., signatures)

This should be set to .950

Click here to start the process

2. Click **Close** in the **Data Preparation** menu to clear it from the screen.

3. In the Unsupervised Classification dialog under **Input Raster File**, enter **germtm.img**. This is the image file that is classified.

4. Under **Output File**, enter **germtm_isodata.img** in the directory of your choice.

   This is the name for the output thematic raster layer.

Set Initial Cluster Options

The Clustering Options allow you to define how the initial clusters are generated.

5. Under **Clustering Options**, enter **10** in the **Number of Classes** field.

Choose Processing Options

The Processing Options allow you to specify how the process is performed.

1. Enter **24** in the **Maximum Iterations** number field under **Processing Options**.

   This is the maximum number of times that the ISODATA utility reclusters the data. It prevents this utility from running too long, or from potentially getting stuck in a cycle without reaching the convergence threshold.

2. Confirm that the **Convergence Threshold** number field is set to **.950**.

Convergence Threshold

The convergence threshold is the maximum percentage of pixels whose cluster assignments can go unchanged between iterations. This threshold prevents the ISODATA utility from running indefinitely.

By specifying a convergence threshold of .95, you are specifying that as soon as 95% or more of the pixels stay in the same cluster between one iteration and the next, the utility should stop processing. In other words, as soon as 5% or fewer of the pixels change clusters between iterations, the utility stops processing.

3. Click **OK** in the Unsupervised Classification dialog to start the classification process. The Unsupervised Classification dialog closes automatically.

   A Job Status dialog displays, indicating the progress of the function.

   NOTE: This process could take up to 15 minutes, depending upon your hardware capabilities.

4. In the Job Status dialog, click **OK** when the process is 100% complete.

5. Proceed to the “Use Unsupervised Classification” section to analyze the classes, so that you can identify and assign class names and colors.

Evaluate Classification

After a classification is performed, you can use a classification overlay or recode the classes to evaluate and test the accuracy of the classification.

Create Classification Overlay

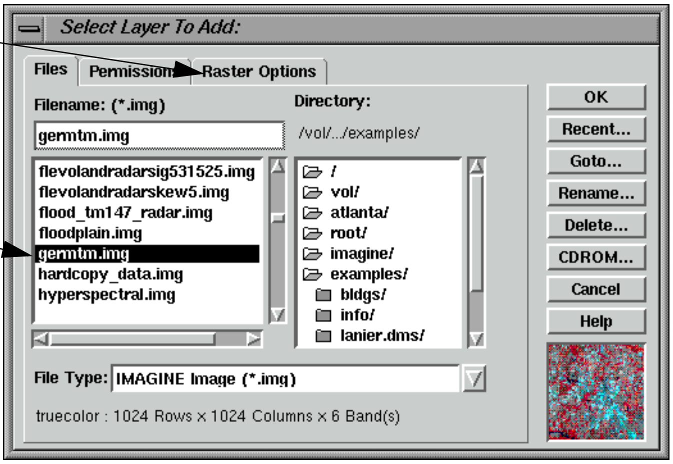
In this example, you use the Raster Attribute Editor to compare the original image data with the individual classes of the thematic raster layer that was created from the unsupervised classification (**germtm_isodata.img**). This process helps identify the classes in the thematic raster layer. You may also use this process to evaluate the classes of a thematic layer that was generated from a supervised classification.

ERDAS IMAGINE should be running and you should have a Viewer open.

Display Files

1. Select **File | Open | Raster Layer** from the Viewer menu bar to display the **germtm.img** continuous raster layer.

   The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
flevolandradarsig531525.img
flevolandradarskew5.img
flood_tm147_radar.img
floodplain.img
germtm.img
hardcopy_data.img
hyperspectral.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) germtm.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

truecolor : 1024 Rows x 1024 Columns x 6 Band(s)

Click on this file tab to see the raster options

Click here to select germtm.img

2. In the Select Layer To Add dialog under **Filename**, select **germtm.img**.
3. Click the Raster Options tab at the top of the Select Layer To Add dialog.
4. Set **Layers to Colors** at **4**, **5**, and **3**, respectively.
5. Click **OK** in the Select Layer To Add dialog to display the image file.
6. Select **File | Open | Raster Layer** from the Viewer menu bar to display the thematic raster layer, **germtm_isodata.img**, over the **germtm.img** file.

   The Select Layer To Add dialog opens.

7. Under **Filename**, open the directory in which you previously saved **germtm_isodata.img** by entering the directory path name in the text entry field and pressing the Return key on your keyboard.

8. Select the file **germtm_isodata.img** from the list of files in the directory.

   You are going to evaluate/identify the classes in this file.

9. Click the Raster Options tab at the top of the Select Layer To Add dialog.

10. Click **Clear Display** to turn off this checkbox.

11. Click **OK** in the Select Layer To Add dialog to display the image file.

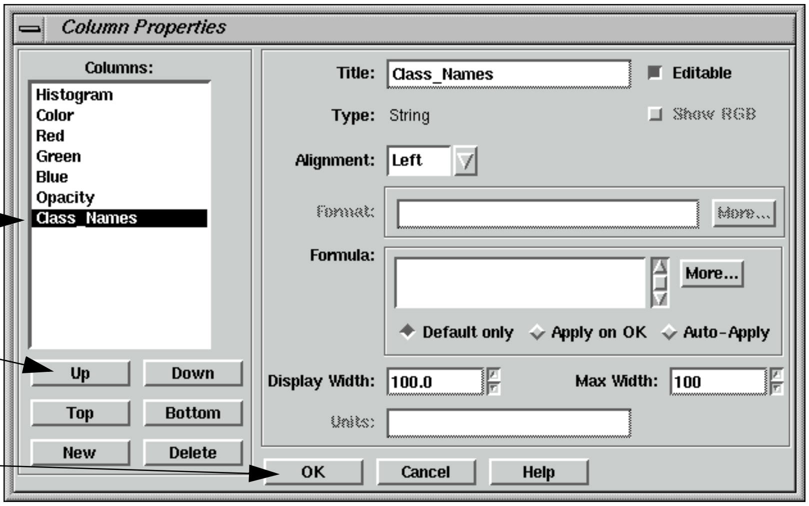
Open Raster Attribute Editor

1. Select **Raster | Attributes** from the Viewer menu bar.

   The Raster Attribute Editor displays.

2. In the Raster Attribute Editor, select **Edit | Column Properties** to rearrange the columns in the CellArray so that they are easier to view.

   The Column Properties dialog opens.



Column Properties

Columns:
Histogram
Color
Red
Green
Blue
Opacity
Class_Names

Title: Class_Names ☑ Editable
Type: String ☐ Show RGB
Alignment: Left
Format:
Formula:
● Default only ◇ Apply on OK ◇ Auto-Apply
Display Width: 100.0
Max Width: 100
Units:

Click here to move this column

Click here to move the selected column up

Click here to rearrange columns

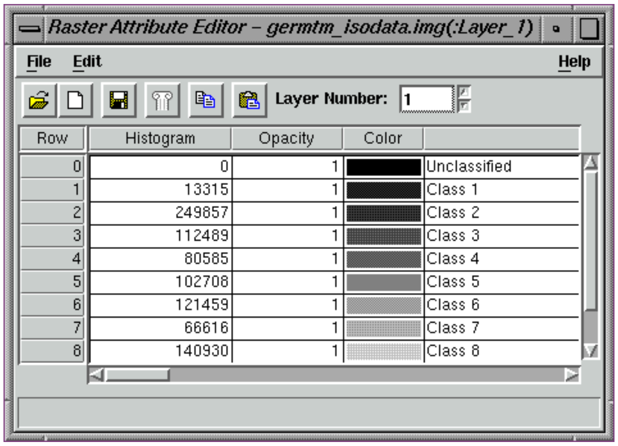
3. In the Column Properties dialog under **Columns**, select **Opacity**, then click **Up** to move **Opacity** so that it is under **Histogram**.

4. Select **Class_Names**, then click **Up** to move **Class_Names** so that it is under **Color**.

5. Click **OK** in the Column Properties dialog to rearrange the columns in the Raster Attribute Editor.

   The Column Properties dialog closes.

   The data in the Raster Attribute Editor CellArray should appear similar to the following example:



Raster Attribute Editor – germtm_isodata.img(:Layer_1)
File
Edit
Help
Layer Number: 1

Row	Histogram	Opacity	Color	
0	0	1	■	Unclassified
1	13315	1	■	Class 1
2	249857	1	■	Class 2
3	112489	1	■	Class 3
4	80585	1	■	Class 4
5	102708	1	■	Class 5
6	121459	1	■	Class 6
7	66616	1	■	Class 7
8	140930	1	■	Class 8

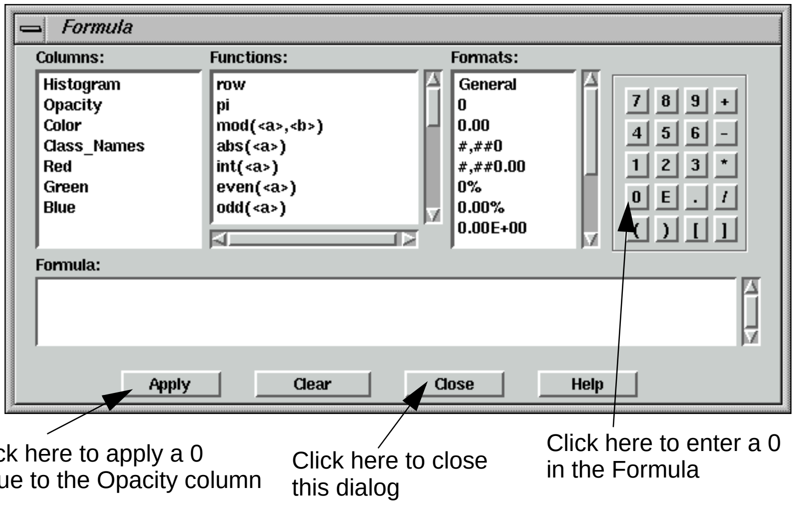
Analyze Individual Classes

Before you can begin to analyze the classes individually, you need to set the opacity for all of the classes to zero.

1. In the Raster Attribute Editor, click on the word **Opacity** at the top of the **Opacity** column to select all of the classes.

2. In the Raster Attribute Editor, right-hold on the word **Opacity** at the top of the **Opacity** column and select **Formula** from the **Column Options** menu.

   The Formula dialog opens.



Formula
Columns:
Histogram
Opacity
Color
Class_Names
Red
Green
Blue
Functions:
row
pi
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
Formats:
General
0
0.00
#,##0
#,##0.00
0%
0.00%
0.00E+00
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
Formula:
Apply
Clear
Close
Help

Click here to apply a 0 value to the Opacity column

Click here to close this dialog

Click here to enter a 0 in the Formula

3. In the Formula dialog, click **0** in the number pad.

   A **0** is placed in the **Formula** field.

4. In the Formula dialog, click **Apply** to change all of the values in the **Opacity** column to **0**, and then click **Close**.

5. In the Raster Attribute Editor, click and hold on the color patch under **Color** for **Class 1** in the CellArray and change the color to **Yellow**. This provides better visibility in the Viewer.

6. Verify the **Opacity** for **Class 1** in the CellArray is set to **1**.

   This class is shown in the Viewer.

7. In the Viewer menu bar, select **Utility | Flicker** to analyze which pixels are assigned to this class.

   The Viewer Flicker dialog opens.

8. Turn on the **Auto Mode** in the Viewer Flicker dialog by clicking on the checkbox.

   The flashing black pixels in the **germtm.img** file are the pixels of this class. These areas are water.

9. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 1**. (You may need to double-click in the column.) Change this name to **Water** and then press Return on the keyboard.

10. In the Raster Attribute Editor, click and hold on the **Color** patch for **Water** and select **Blue** from the popup list. (You may need to select the entire row for this class first.)

11. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Water** back to **0** in the Raster Attribute Editor. Press Return on the keyboard.

12. Change the **Color** for **Class 2** in the CellArray to **Yellow** for better visibility in the Viewer.

13. Change the **Opacity** for **Class 2** to **1** and press Return on the keyboard.

    This class is shown in the Viewer.

14. In the Viewer menu bar, select **Utility | Flicker** to analyze which pixels are assigned to this class.

    The Viewer Flicker dialog opens.

15. Turn on the **Auto Mode** in the Viewer Flicker dialog.

    The flashing yellow pixels in the **germtm.img** file should be the pixels of this class. These are forest areas.

16. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 2**. (You may need to double-click in the column.) Change this name to **Forest**, then press Return on the keyboard.

17. In the Raster Attribute Editor, click and hold on the **Color** patch for **Forest** and select **Pink** from the popup list. (You may need to select the entire row for this class first.)

18. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Forest** back to **0**. Press Return on the keyboard.

19. Repeat these steps with each class so that you can see how the pixels are assigned to each class. You may also try selecting more than one class at a time.

20. Continue assigning names and colors for the remaining classes in the Raster Attribute Editor CellArray.

21. In the Raster Attribute Editor, select **File | Save** to save the data in the CellArray.

22. Select **File | Close** from the Raster Attribute Editor menu bar.

23. Select **File | Clear** from the Viewer menu bar.

CHAPTER SIX

Polynomial Rectification

Introduction

Rectification is the process of projecting the data onto a plane and making it conform to a map projection system. Assigning map coordinates to the image data is called georeferencing. Since all map projection systems are associated with map coordinates, rectification involves georeferencing.

Approximate completion time for this tour guide is 1 hour.

Rectify a Landsat Image

Perform Image to Image Rectification

In this tour guide, you rectify a Landsat TM image of Atlanta, Georgia, using a georeferenced SPOT panchromatic image of the same area. The SPOT image is rectified to the State Plane map projection.

In rectifying the Landsat image, you use these basic steps:

- display files
- start Geometric Correction Tool
- record GCPs
- compute a transformation matrix
- resample the image
- verify the rectification process

Display Files

First, you display the image to be rectified and an image that is already georeferenced.

ERDAS IMAGINE must be running and a Viewer open.

1. Click the Viewer icon on the ERDAS IMAGINE icon panel to open a second Viewer.

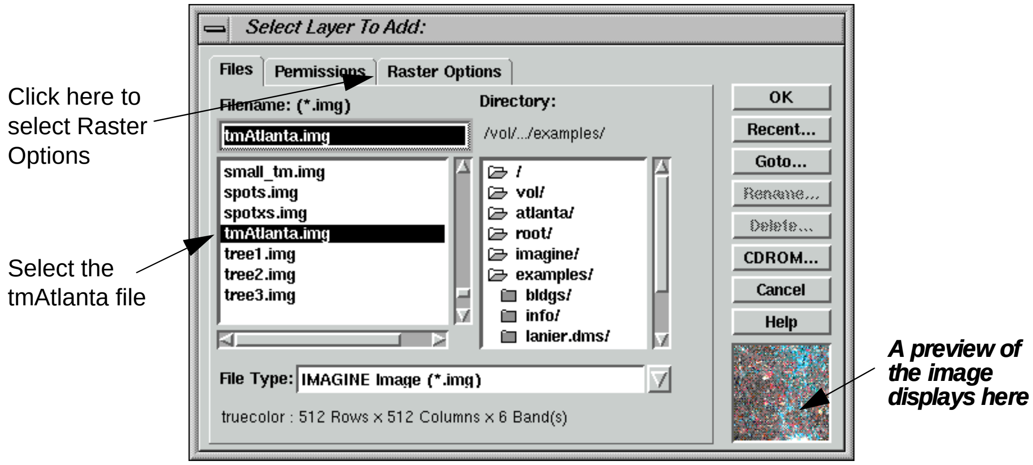
Viewer

The second Viewer displays on top of the first Viewer.

2. In the ERDAS IMAGINE menu bar, select **Session | Tile Viewers** to position the Viewers side by side.

3. In the first Viewer’s tool bar, click the Open icon (or select **File | Open | Raster Layer**).

The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
small_tm.img
spots.img
spotxs.img
tmAtlanta.img
tree1.img
tree2.img
tree3.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/

Filename: (*.img) tmAtlanta.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

truecolor : 512 Rows x 512 Columns x 6 Band(s)

Click here to select Raster Options

Select the tmAtlanta file

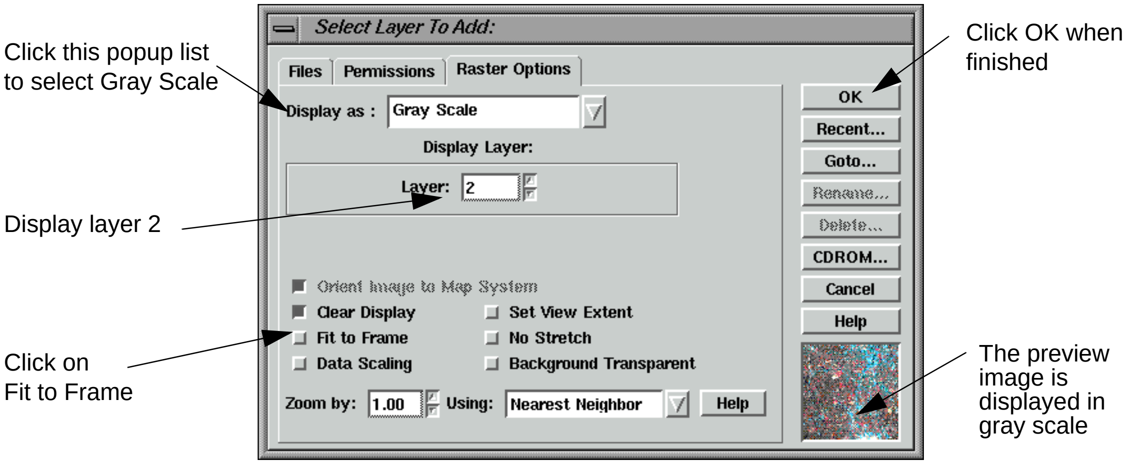
A preview of the image displays here

4. In the Select Layer To Add dialog under **Filename**, click on the file **tmAtlanta.img**.

This file is a Landsat TM image of Atlanta. This image has not been rectified.

5. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

The Raster Options display in the Select Layer To Add dialog.


Select Layer To Add:
Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
Orient Image to Map System
Clear Display
Set View Extent
Fit to Frame
No Stretch
Data Scaling
Background Transparent
Help

Display as : Gray Scale

Display Layer:

Layer: 2

Zoom by: 1.00

Using: Nearest Neighbor

Click this popup list to select Gray Scale

Display layer 2

Click on Fit to Frame

Click OK when finished

The preview image is displayed in gray scale

6. Click the **Display as** popup list and select **Gray Scale**.

7. Under **Display Layer**, enter **2**.

   Depending upon your application, it may be easier to select GCPs from a single band of imagery. The image **tmAtlanta.img** displays in **True Color** by default.

8. Click **Fit to Frame**, so that the entire image is visible in the Viewer.

9. Click **OK** in the Select Layer To Add dialog.

   The file **tmAtlanta.img** displays in the first Viewer.

10. In the second Viewer tool bar, click the Open icon (or select **File | Open | Raster Layer**).

    The Select Layer To Add dialog opens.

11. In the Select Layer To Add dialog, click on the file **panAtlanta.img**.

    This file is a SPOT panchromatic image of Atlanta. This image has been georeferenced to the State Plane map projection.

12. Click **OK** in the Select Layer To Add dialog.

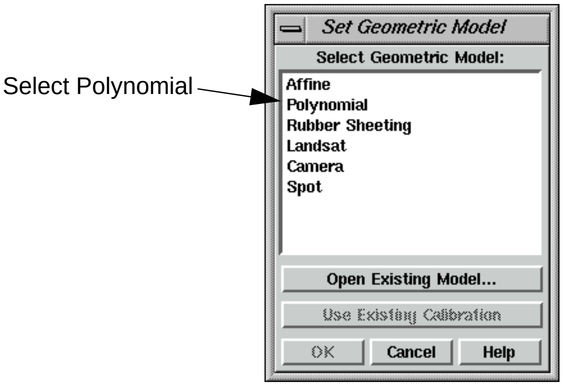
    The file **panAtlanta.img** displays in the second Viewer.

Start GCP Tool

You start the Geometric Correction Tool from the first Viewer—the Viewer displaying the file to be rectified (**tmAtlanta.img**).

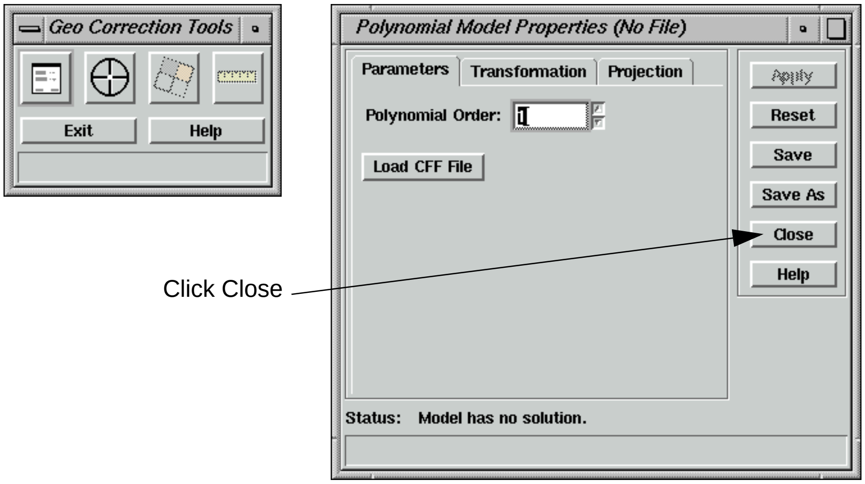
1. Select **Raster | Geometric Correction** from the first Viewer’s menu bar.

   The Set Geometric Model dialog opens.


Select Polynomial
Set Geometric Model
Select Geometric Model:
Affine
Polynomial
Rubber Sheeting
Landsat
Camera
Spot
Open Existing Model...
Use Existing Calibration
OK
Cancel
Help

2. In the Set Geometric Model dialog, select **Polynomial** and then click **OK**.

   The Geo Correction Tools open, along with the Polynomial Model Properties dialog.

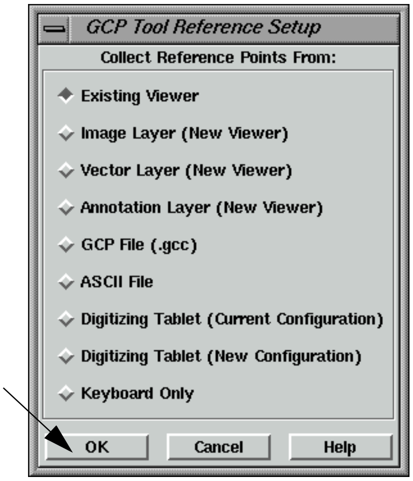

Geo Correction Tools
Exit
Help
Polynomial Model Properties (No File)
Parameters
Transformation
Projection
Load CFF File
Apply
Reset
Save
Save As
Close
Help
Click Close

Polynomial Order: 1

Status: Model has no solution.

3. Click **Close** in the Polynomial Model Properties dialog to close it for now. You select these parameters later.

   The GCP Tool Reference Setup dialog opens.


GCP Tool Reference Setup
Collect Reference Points From:
◆ Existing Viewer
◇ Image Layer (New Viewer)
◇ Vector Layer (New Viewer)
◇ Annotation Layer (New Viewer)
◇ GCP File (.gcc)
◇ ASCII File
◇ Digitizing Tablet (Current Configuration)
◇ Digitizing Tablet (New Configuration)
◇ Keyboard Only
OK
Cancel
Help

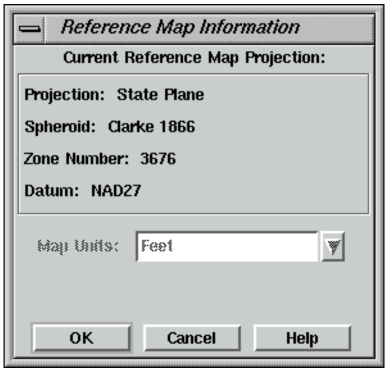
Click OK to
accept the default

4. Accept the default of **Existing Viewer** in the GCP Tool Reference Setup dialog by clicking **OK**.

   The GCP Tool Reference Setup dialog closes and a Viewer Selection Instructions box opens, directing you to click in a Viewer to select for reference coordinates.

5. Click in the second Viewer, which displays **panAtlanta.img**.

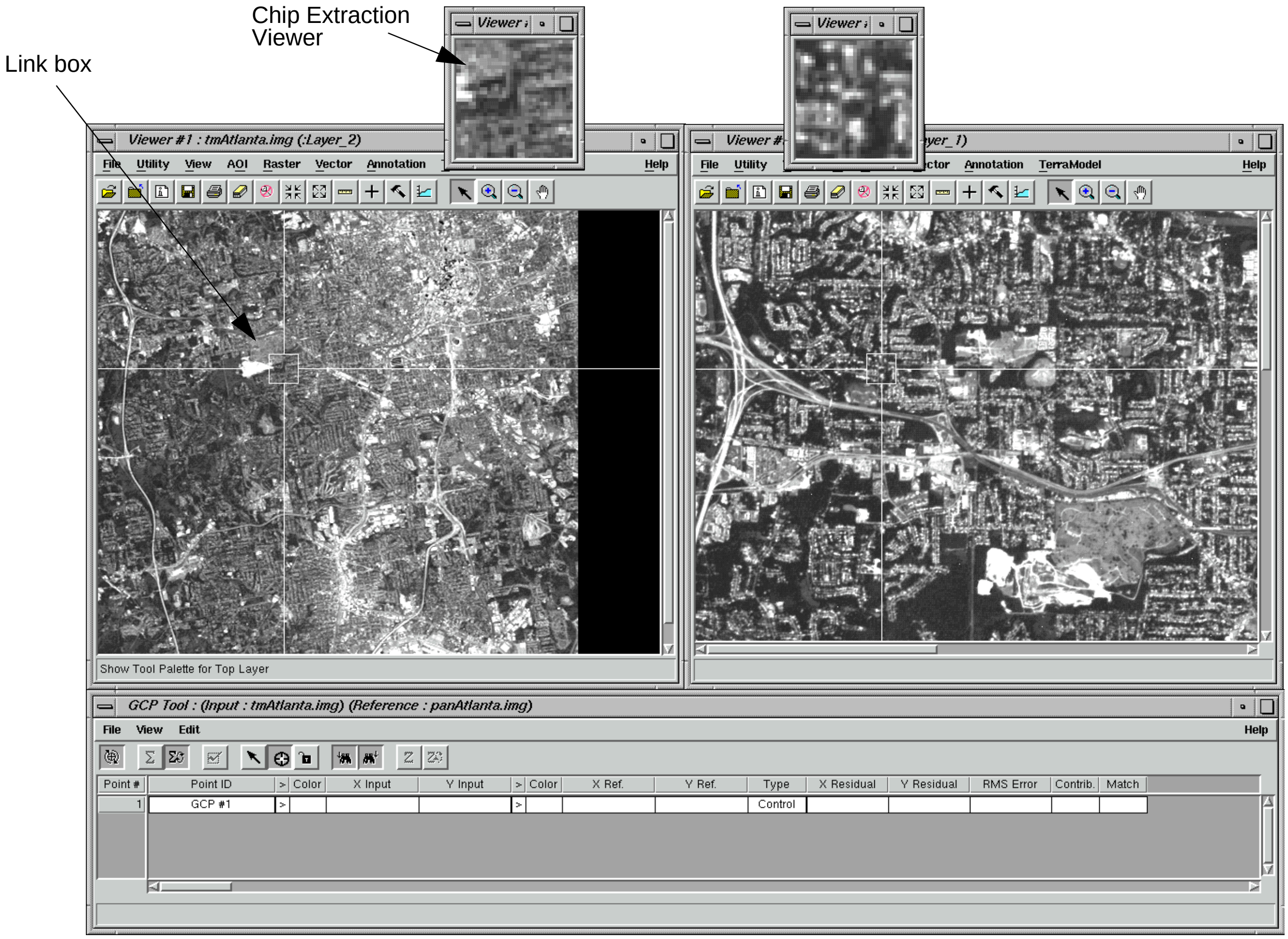
   The Reference Map Information dialog opens showing the map information for the georeferenced image. The information in this dialog is not editable.


Reference Map Information
Current Reference Map Projection:
OK
Cancel
Help

Projection: State Plane
Spheroid: Clarke 1866
Zone Number: 3676
Datum: NAD27
Map Units: Feet

6. Click **OK** in the Reference Map Information dialog.

   The Chip Extraction Viewers (Viewers #3 and #4), link boxes, and the GCP Tool open. The link boxes and GCP Tool are automatically arranged on the screen (you can turn off this option in the ERDAS IMAGINE Preferences). You may want to resize and move the link boxes so that they are easier to see.


Chip Extraction Viewer
Link box
Viewer #1 : tmAtlanta.img (:Layer_2)
File Utility View AOI Raster Vector Annotation Help
Show Tool Palette for Top Layer
Viewer #
ector Annotation TerraModel Help
GCP Tool : (Input : tmAtlanta.img) (Reference : panAtlanta.img)
File View Edit Help

Point #	Point ID	>	Color	X Input	Y Input	>	Color	X Ref.	Y Ref.	Type	X Residual	Y Residual	RMS Error	Contrib.	Match
1	GCP #1	>				>				Control					

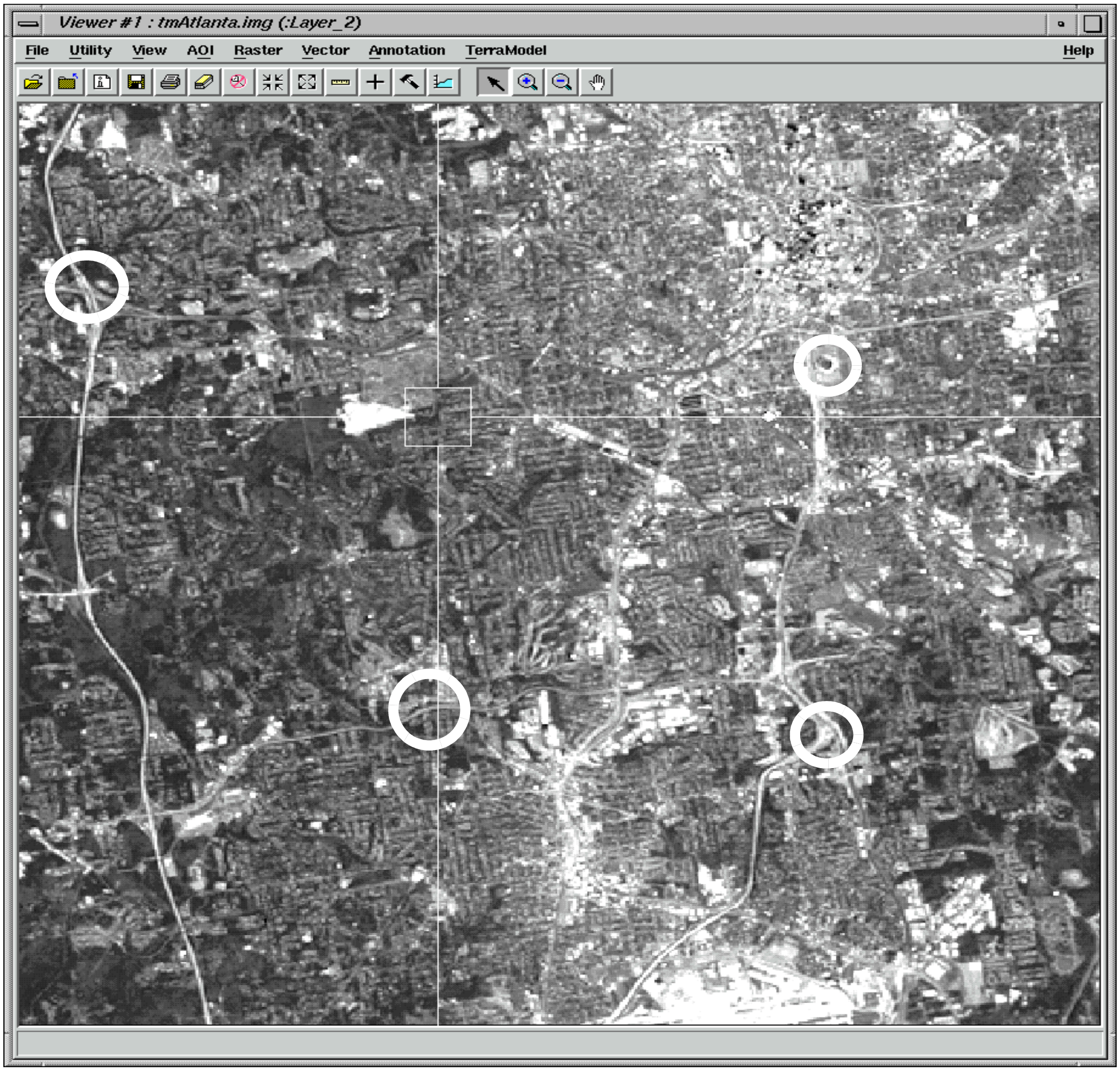
In this tour guide, you are going to rectify **tmAtlanta.img** in the first Viewer to **panAtlanta.img** in the second Viewer.

Select GCPs

When the GCP Tool is started, the tool is set in Automatic GCP Editing mode by default.

The following icon is active, indicating that this is the case.

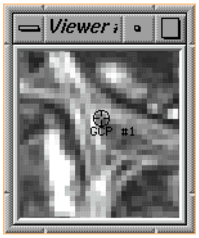
1. In the first Viewer, select one of the areas shown in the following picture by clicking on that area. The circled areas are locations for GCPs. You should choose points that are easily identifiable in both images, such as road intersections and landmarks.


Viewer #1 : tmAtlanta.img (:Layer_2)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

The point you have selected is marked as GCP #1 in the Viewer and its X and Y inputs are listed in the GCP Tool CellArray.

2. In order to make GCP #1 easier to see, right-hold in the **Color** column to the right of GCP #1 in the GCP Tool CellArray and select the color **Black**.

3. In Viewer #3 (the Chip Extraction Viewer associated with the first Viewer), drag the GCP to the exact location you would like it to be.



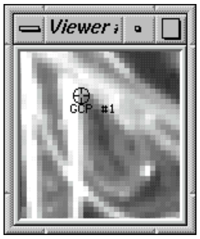
NOTE: (UNIX only) To view the GCP while you are dragging it, turn off the **Use Fast Selectors** *checkbox in the* **Viewer** *category under* **Session | Preferences** *(this change does not take effect until the Viewer is restarted).*

4. In the GCP Tool, click on the Create GCP icon.

⌖

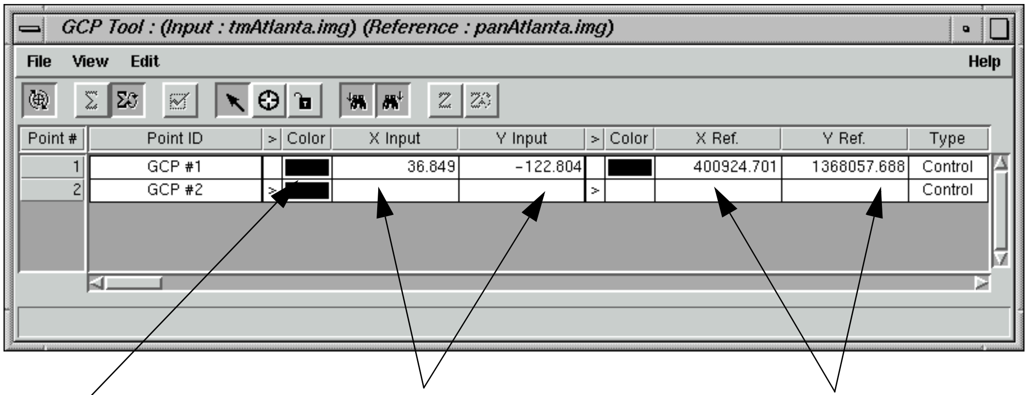
5. In the second Viewer, click in the same area that is covered in the source Chip Extraction Viewer (Viewer #3).

The point you have selected is marked as GCP #1 in the Viewer, and its X and Y coordinates are listed in the GCP Tool CellArray.



6. In order to make GCP #1 easier to see in the second Viewer, right-hold in the **Color** column to the left of the X reference for GCP #1 in the GCP Tool CellArray and select the color **Black**.

The GCP Tool should now look similar to the one pictured below:


GCP Tool : (Input : tmAtlanta.img) (Reference : panAtlanta.img)
File
View
Edit
Help

Point #	Point ID	>	Color	X Input	Y Input	>	Color	X Ref.	Y Ref.	Type
1	GCP #1		■	36.849	-122.804		■	400924.701	1368057.688	Control
2	GCP #2	>	■			>				Control

Select the color for the source GCP here

These are the X and Y file coordinates for GCPs in the input image (tmAtlanta.img)

These are the X and Y map coordinates for GCPs in the reference image (panAtlanta.img)

7. In Viewer #4 (the Chip Extraction Viewer associated with the second Viewer), drag the GCP to the same location you moved it to in Viewer #3.

8. Click on the Create GCP icon in the GCP tool bar.

9. Return to the source Viewer (the first Viewer) and click to digitize another GCP.

10. In order to make GCP #2 easier to see, right-hold in the **Color** column to the right of GCP #2 in the GCP Tool CellArray and select the color **Magenta**.

11. In Viewer #3, drag the new GCP (GCP #2) to the exact location you would like it to be.

12. Repeat steps (4. and 5. to digitize the same point in the second Viewer.

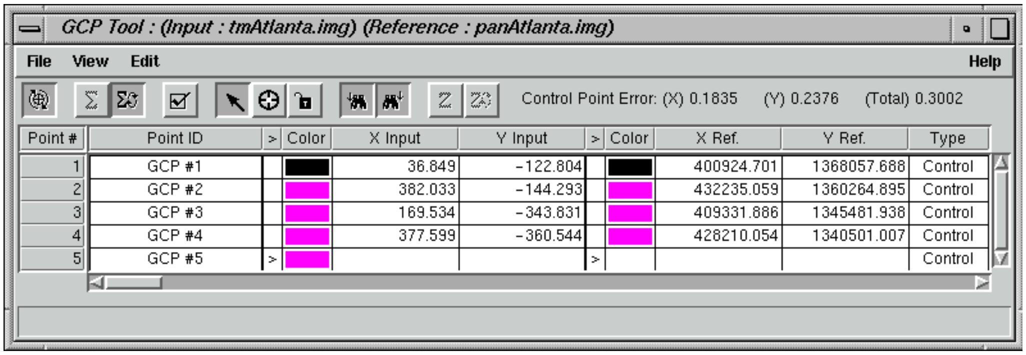
13. As in step 10., you can change the color of the GCP marker to make it easier to see.

14. Digitize at least two more GCPs in each Viewer (on **tmAtlanta.img** in the first Viewer and **panAtlanta.img** in the second Viewer) by repeating the above steps. The GCPs you digitize should be spread out across the image to form a large triangle (i.e., they should not form a line).

15. Choose colors that enable you to see the GCPs in the Viewers.

After you digitize the fourth GCP in the first Viewer, note that the GCP is automatically matched in the second Viewer. This occurs with all subsequent GCPs that you digitize.

After you digitize GCPs in the Viewers, the GCP Tool CellArray should look similar to the following example:


GCP Tool : (Input : tmAtlanta.img) (Reference : panAtlanta.img)
File
View
Edit
Help

Control Point Error: (X) 0.1835 (Y) 0.2376 (Total) 0.3002

Point #	Point ID	>	Color	X Input	Y Input	>	Color	X Ref.	Y Ref.	Type
1	GCP #1		■	36.849	-122.804		■	400924.701	1368057.688	Control
2	GCP #2		🟪	382.033	-144.293		🟪	432235.059	1360264.895	Control
3	GCP #3		🟪	169.534	-343.831		🟪	409331.886	1345481.938	Control
4	GCP #4		🟪	377.599	-360.544		🟪	428210.054	1340501.007	Control
5	GCP #5	>	🟪			>				Control

Selecting GCPs

Selecting GCPs is useful for moving GCPs graphically or deleting them. You can select GCPs graphically (in the Viewer) or in the GCP CellArray.

- To select a GCP graphically in the Viewer, use the Select icon.

  Select it as you would an annotation element. When a GCP is selected, you can drag it to move it to the desired location.

  You can also click on any GCP coordinate in the CellArray to enter new coordinates.

- To select GCPs in the CellArray, click in the **Point #** column, or use any of the CellArray selection options in the right mouse button menu (right-hold in the **Point #** column).

Deleting a GCP

To delete a GCP, select the GCP in the CellArray in the GCP Tool and then right-hold in the **Point #** column to select **Delete Selection**.

Compute Transformation Matrix

A transformation matrix is a set of numbers that can be plugged into polynomial equations. These numbers are called the transformation coefficients. The polynomial equations are used to transform the coordinates from one system to another.

The **Transformation** tab in the Polynomial Model Properties dialog shows you a scrolling list of the transformation coefficients arranged in the transformation matrix. To access the Polynomial Model Properties dialog and the **Transformation** tab, click the Display Model Properties icon in the Geo Correction Tools.

The coefficients are placed in the transform editor in two ways:

- the **Transformation** tab CellArray is automatically populated when the model is solved in the GCP Tool
- using the CellArray located in the GCP tool to enter them directly from the keyboard.

In this tour guide, the transformation coefficients are calculated from the GCP Tool, and are automatically recorded in the **Transformation** tab.

Preparation

A minimum number of GCPs is necessary to calculate the transformation, depending on the order of the transformation. This number of points is:

$$\frac{(t+1)(t+2)}{2}$$

Where t is the order of the transformation.

If the minimum number of points is not satisfied, then a message displays notifying you of that condition, and the RMS errors and residuals are blank. At this point, you are not allowed to resample the data.

Change Order of Transformation

To change the order of the transformation, use the Polynomial Model Properties dialog (available from the Geo Correction Tools). Using this dialog, select the **Parameters** tab at the top of the dialog. This tab allows the polynomial order to be altered.

Calculate Transformation Matrix from GCPs

The Auto Calculation function is enabled by default in the GCP Tool. The Auto Calculation function computes the transformation in real time as you edit the GCPs or change the selection in the CellArray.

With the Automatic Transform Calculation tool activated, you can move a GCP in the Viewer while watching the transformation coefficients and errors change at the top of the GCP Tool.

You may want to turn off the Auto Calculation function if your system or computation is taking too long.

NOTE: Some models do not support Auto Calculation. If this is the case, the function is disabled.

1. If your model does not support Auto Calculation, click on the Calculate icon in the GCP Tool tool bar.

Σ

NOTE: The transformation matrix contains the coefficients for transforming the reference coordinate system to the input coordinate system. Therefore, the units of the residuals and RMS errors are the units of the input coordinate system. In this tour guide, the input coordinate system is pixels.

Digitize Check Points

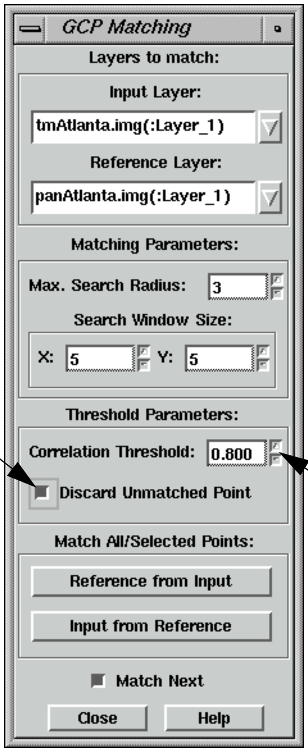
Check points are useful in independently checking the accuracy of your transformation.

2. In the GCP Tool, turn all of the GCPs to yellow by right-holding **Select All** in the **Point #** column and then right-holding **Yellow** in each of the two **Color** columns.
3. Right-hold **Select None** in the **Point #** column of the GCP Tool CellArray to deselect the GCPs.
4. In the last row of the CellArray, right-hold in each of the two **Color** columns and select **Magenta**.

   All of the check points you add in the next steps are Magenta, which distinguishes them from the GCPs.
5. Select the last row of the CellArray by clicking in the **Point #** column next to that row.
6. Select **Edit | Set Point Type | Check** from the GCP Tool menu bar.

   All of the points you add in the next steps are classified as check points.
7. Select **Edit | Point Matching** from the GCP Tool menu bar.

   The GCP Matching dialog opens.



GCP Matching

Layers to match:

Input Layer: tmAtlanta.img(:Layer_1)

Reference Layer: panAtlanta.img(:Layer_1)

Matching Parameters:

Max. Search Radius: 3

Search Window Size: X: 5 Y: 5

Threshold Parameters:

Correlation Threshold: 0.800

☐ Discard Unmatched Point

Click here to activate this checkbox

Set the Correlation Threshold here

8. In the GCP Matching dialog under **Threshold Parameters**, change the **Correlation Threshold** to **.8**, and then press the Return key on your keyboard.

9. Click the **Discard Unmatched Point** checkbox to activate it.

10. Click **Close** in the GCP Matching dialog.

11. In the GCP Tool, click the Create GCP icon and then the Lock icon.

12. Create 5 check points in each of the two Viewers, just as you did for the GCPs.

    NOTE: If the previously input points were not accurate, then the check points you designate may go unmatched and be automatically discarded.

13. When the 5 check points have been created, click the Lock icon in the GCP Tool to unlock the Create GCP function.

14. Click the Compute Error icon in the GCP Tool to compute the error for the check points.

    The **Check Point Error** displays at the top of the GCP Tool. A total error of less than 1 pixel error would make it a reasonable resampling.

15. To view the polynomial coefficients, click the Model Properties icon in the Geo Correction Tools.

The Polynomial Model Properties dialog opens.

16. Once you have checked the tabs of the Polynomial Model Properties dialog, click **Close** in the Polynomial Properties dialog.

Resample the Image

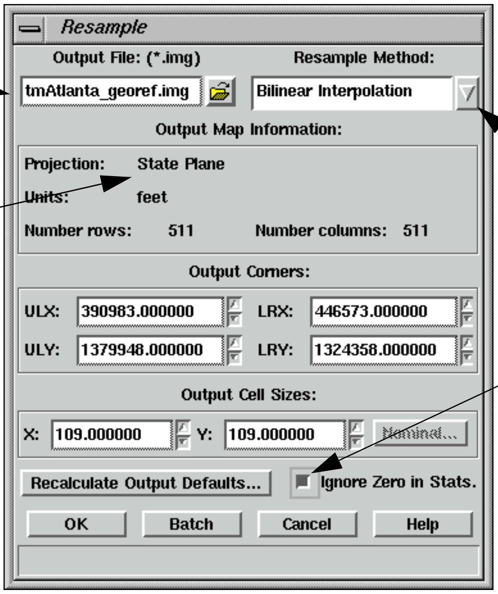
Resampling is the process of calculating the file values for the rectified image and creating the new file. All of the raster data layers in the source file are resampled. The output image has as many layers as the input image.

ERDAS IMAGINE provides these widely-known resampling algorithms:Nearest Neighbor, Bilinear Interpolation, Cubic Convolution, and Bicubic Spline.

Resampling requires an input file and a transformation matrix by which to create the new pixel grid.

1. Click on the Resample icon in the Geo Correction Tools.

The Resample dialog opens.



Output File: (*.img) tmAtlanta_georef.img

Resample Method: Bilinear Interpolation

Output Map Information:

Projection: State Plane

Units: feet

Number rows: 511 Number columns: 511

Output Corners:

ULX: 390983.000000 LRX: 446573.000000

ULY: 1379948.000000 LRY: 1324358.000000

Output Cell Sizes:

X: 109.000000 Y: 109.000000 Nominal...

Recalculate Output Defaults... ☑ Ignore Zero in Stats.

Enter file name for the new georeferenced image file

Output map projection should be State Plane

Click here to select the Bilinear Interpolation resampling method

Click here to exclude zero file values in statistics for output file

2. In the Resample dialog under **Output File**, enter the name **tmAtlanta_georef.img** for the new resampled data file. This is the output file from rectifying the **tmAtlanta.img** file to the coordinate system of the **panAtlanta.img** file.

   NOTE: Be sure to enter the output file in a directory where you have write permission and at least 25 Mb of free disk space.

3. Under **Resample Method**, click the popup list and select **Bilinear Interpolation**.

4. Click **Ignore Zero in Stats.**, so that pixels with zero file values are excluded when statistics are calculated for the output file.

5. Click **OK** in the Resample dialog to start the resampling process.

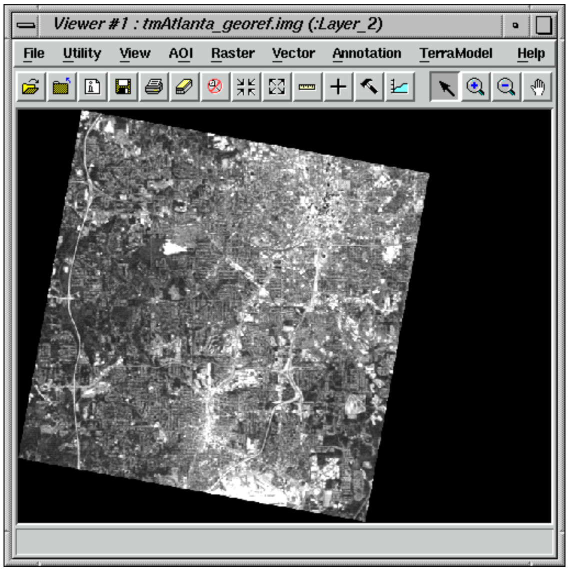
   A Job Status dialog opens to let you know when the processes complete.

6. Click **OK** in the Job Status dialog when the job is 100% complete.

Verify the Rectification Process

One way to verify that the input image (**tmAtlanta.img**) has been correctly rectified to the reference image (**panAtlanta.img**) is to display the resampled image (**tmAtlanta_georef.img**) and the reference image and then visually check that they conform to each other.

1. Display the resampled image (**tmAtlanta_georef.img**) in the first Viewer. Use the **Clear Display** option in the Select Layer To Add dialog to remove **tmAtlanta.img** from the Viewer before the resampled image is opened.



2. When **tmAtlanta.img** closes in the first Viewer, you are asked if you want to save your changes. Click **No** in all of the Save Changes dialogs.

   The Geometric Correction Tool exits.

3. Right-hold **Geo. Link/Unlink** under the **Quick View** menu in the first Viewer.

4. Click in the second Viewer to link the Viewers together.

5. Right-hold **Inquire Cursor** under the **Quick View** menu in the first Viewer.

   The inquire cursor (a crosshair) is placed in both Viewers. An Inquire Cursor dialog also opens.

6. Drag the inquire cursor around to verify that it is in approximately the same place in both Viewers. Notice that, as the inquire cursor is moved, the data in the Inquire Cursor dialog are updated.

7. When you are finished, click **Close** in the Inquire Cursor dialog.

Rotate, Flip, or Stretch Images

It is often necessary to perform a first-order rectification to a layer displayed in the Viewer. You may need to rotate, flip, or stretch the image so that North is up.

Choose Model Properties

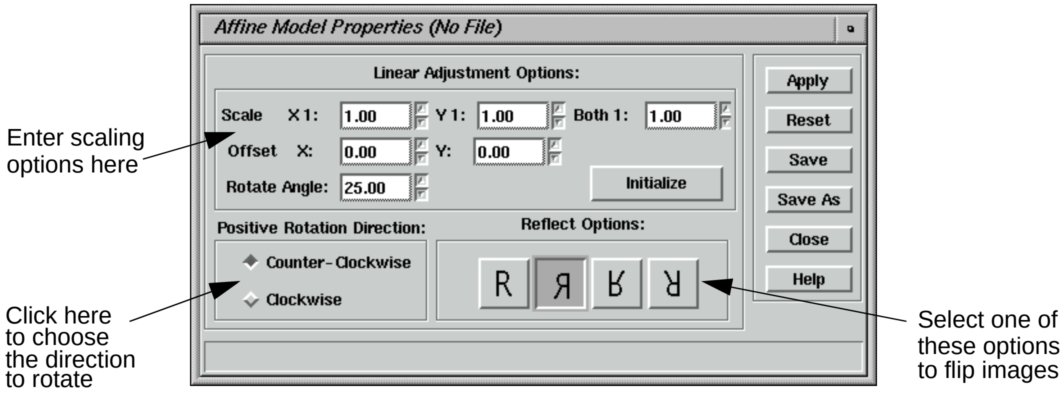
1. Display the file **tmAtlanta.img** in a Viewer.

2. In the Viewer, select **Raster | Geometric Correction**.

   The Set Geometric Model dialog opens.

3. In the Set Geometric Model dialog, click **Affine** and then **OK**.

   The Geo Correction Tools open, along with the Affine Model Properties dialog.


Affine Model Properties (No File)

Linear Adjustment Options:

Scale X1: 1.00 Y1: 1.00 Both 1: 1.00

Offset X: 0.00 Y: 0.00

Rotate Angle: 25.00

Positive Rotation Direction:

◆ Counter-Clockwise

◇ Clockwise

Enter scaling options here

Click here to choose the direction to rotate

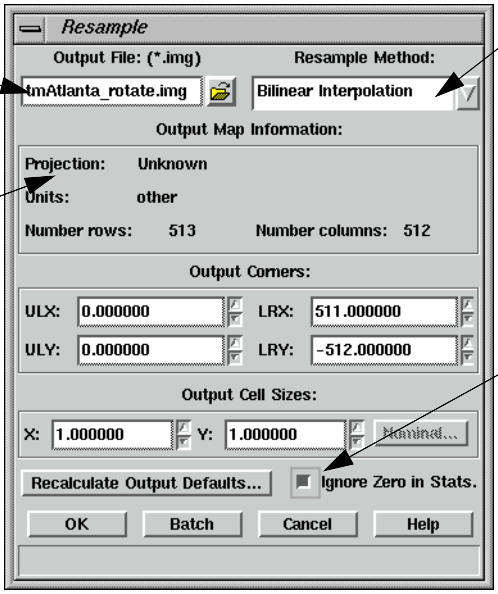
Select one of these options to flip images

4. Change the **Rotate Angle** to **25**.

5. Select the desired **Reflect Option** in the Affine Model Properties dialog, then click **Apply** and **Close**.

6. Click on the Resample icon in the Geo Correction Tools.

The Resample dialog opens.



Output File: (*.img) tmAtlanta_rotate.img

Resample Method: Bilinear Interpolation

Output Map Information:

Projection: Unknown

Units: other

Number rows: 513 Number columns: 512

Output Corners:

ULX: 0.000000 LRX: 511.000000

ULY: 0.000000 LRY: -512.000000

Output Cell Sizes:

X: 1.000000 Y: 1.000000

Enter file name for the new georeferenced image file

Output map projection is Unknown

Click here to select the Bilinear Interpolation resampling method

Click here to exclude zero file values in statistics for output file

7. In the Resample dialog under **Output File**, enter the name **tmAtlanta_rotate.img**.

8. Under **Resample Method**, click the popup list and select **Bilinear Interpolation**.

9. Click **Ignore Zero in Stats.**, so that pixels with zero file values are excluded when statistics are calculated for the output file.

10. Click **OK** in the Resample dialog to start the resampling process.

    A Job Status dialog opens to let you know when the processes complete.

11. Click **OK** in the Job Status dialog when the job is 100% complete.

Check Results

1. Open a new Viewer.

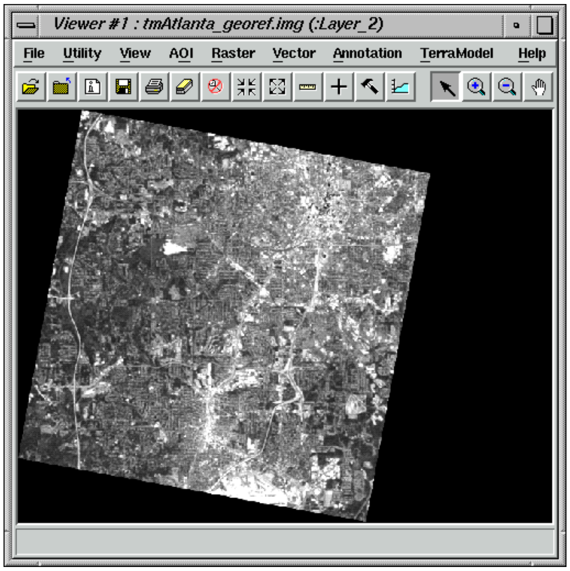
2. Click the Open icon, then select **tmAtlanta_rotate.img** from the directory in which you saved it.

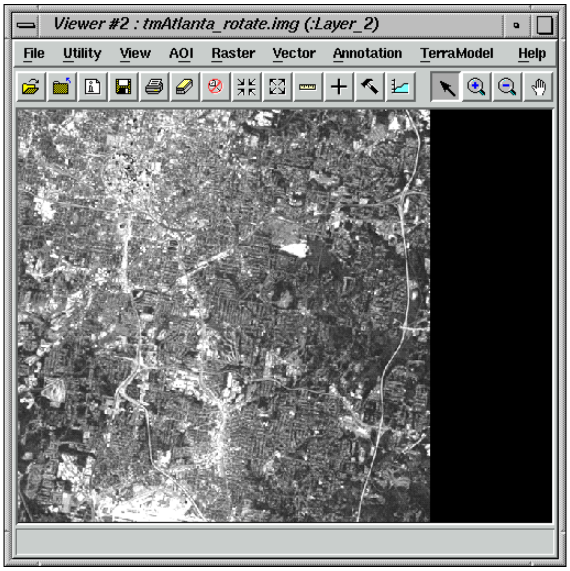
3. Click the **Raster Options** tab, and click the **Display as** dropdown list to select **Gray Scale**.

4. In the **Display Layer** section, select **Layer 2**.

5. Click **OK** in the Select Layer To Add dialog.

6. Compare **tmAtlanta_georef.img** and **tmAtlanta_rotate.img** side by side.


Viewer #1 : tmAtlanta_georef.img (:Layer_2)
File Utility View AOI Raster Vector Annotation TerraModel Help


Viewer #2 : tmAtlanta_rotate.img (:Layer_2)
File Utility View AOI Raster Vector Annotation TerraModel Help

CHAPTER SEVEN

Image Commands

Introduction

ERDAS IMAGINE gives you access to a tool called the Image Commands tool. With it, you can take any file supported by ERDAS IMAGINE and perform many types of operations. One such operation is using the Image Commands tool to create a world file. You can then use the world file with other software packages, such as ESRI's ArcView.

In this tour guide, you can learn how to:

- access the Image Commands tool
- create a world file from a .tif file

Approximate completion time for this tour guide is 15 minutes.

Image Commands

With the Image Commands tool, you can make many changes to your files.

Use Image Interpreter Utilities

ERDAS IMAGINE must be running

1. Click the Interpreter icon on the ERDAS IMAGINE icon panel.

   The **Image Interpreter** menu opens.

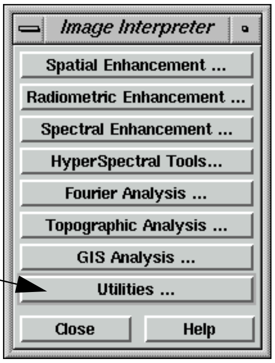
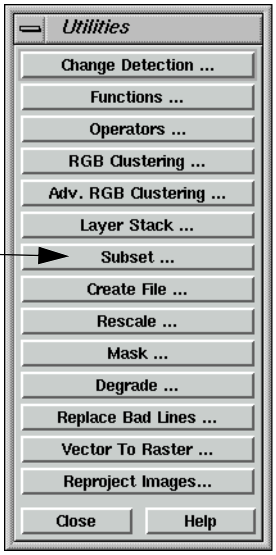


Image Interpreter
Spatial Enhancement ...
Radiometric Enhancement ...
Spectral Enhancement ...
HyperSpectral Tools...
Fourier Analysis ...
Topographic Analysis ...
GIS Analysis ...
Utilities ...
Close
Help

Click the Utilities option

2. Click **Utilities** in the **Image Interpreter** menu.

   The **Utilities** menu opens.

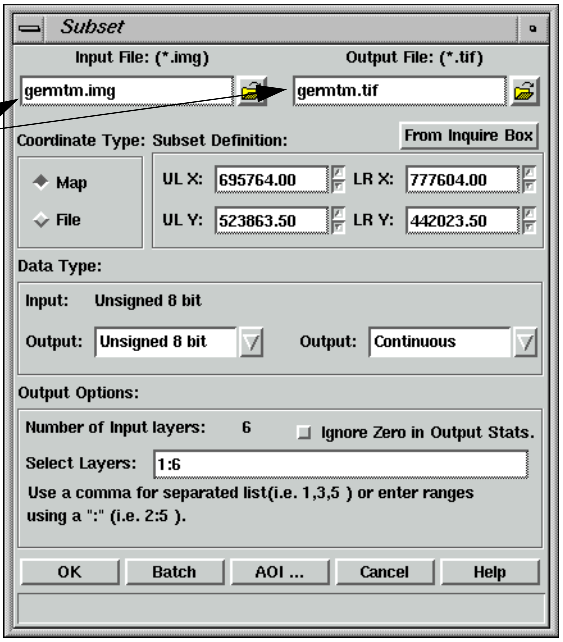

Utilities
Change Detection ...
Functions ...
Operators ...
RGB Clustering ...
Adv. RGB Clustering ...
Layer Stack ...
Subset ...
Create File ...
Rescale ...
Mask ...
Degrade ...
Replace Bad Lines ...
Vector To Raster ...
Reproject Images...
Close
Help

Select the Subset option to create a .tif file

Use the Subset Function

1. In the **Utilities** menu, select **Subset**.

   The Subset dialog opens.


Subset
From Inquire Box
OK
Batch
AOI ...
Cancel
Help

Input File: (*.img) germtm.img

Output File: (*.tif) germtm.tif

Coordinate Type: ● Map ○ File

Subset Definition:

UL X: 695764.00 LR X: 777604.00

UL Y: 523863.50 LR Y: 442023.50

Data Type:

Input: Unsigned 8 bit

Output: Unsigned 8 bit Output: Continuous

Output Options:

Number of Input layers: 6 ☐ Ignore Zero in Output Stats.

Select Layers: 1:6

Use a comma for separated list(i.e. 1,3,5) or enter ranges using a ":" (i.e. 2:5).

Type the input file here, then select the .tif file type for the output file

2. In the Subset dialog, click the Open icon below **Input File**.

3. In the Input File dialog, navigate to the <IMAGINE_HOME>/examples directory, and select the file **germtm.img**.

4. Click **OK** in the Input File dialog to transfer **germtm.img** to the Subset dialog.

5. Click the Open icon under **Output File**.

6. In the Output File dialog, navigate to a directory where you have write permission.

7. Click the **File Type** dropdown list and choose **TIFF**.

8. Type the name **germtm** in the **Filename** window, then press Return on your keyboard.

   The .tif extension is automatically added. By using the Subset utility in this fashion, you can quickly create a TIFF image from an image file.

9. Click **OK** in the Output File dialog.

   The Subset file updates accordingly.

10. Click **OK** in the Subset dialog to generate **germtm.tif**.

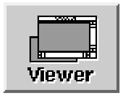
    A Job Status dialog opens, tracking the progress of the function.

11. When the Job is complete, click **OK** in the Job Status dialog.

You can set a preference in the **User Interface & Session** *category to automatically dismiss the Job Status dialog once a job is complete.*

Check the TIFF file

1. Click the Viewer icon on the ERDAS IMAGINE icon panel.



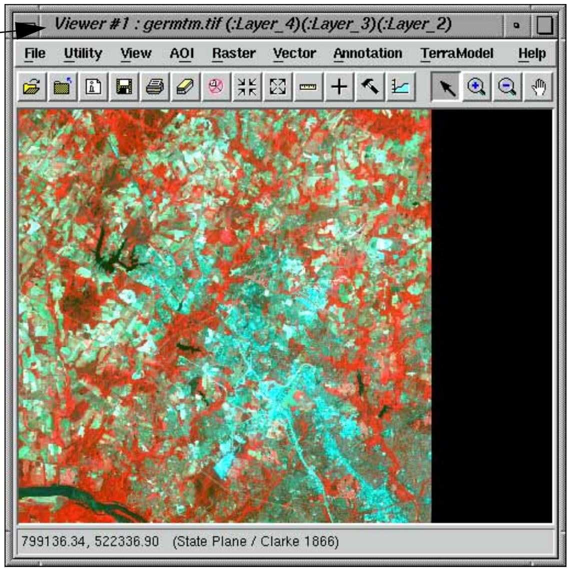
2. Click the Open icon, and click the **Recent** button in the Select Layer To Add dialog.

3. Select the file you just created, **germtm.tif**.

4. Click the **Raster Options** tab, and select **Fit to Frame**.

5. Click **OK** in the Select Layer to Add dialog.

   The TIFF image displays in the Viewer. Notice that it is in the State Plane projection, indicated in the status area of the Viewer.


Viewer #1 : germtm.tif (:Layer_4)(:Layer_3)(:Layer_2)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
799136.34, 522336.90 (State Plane / Clarke 1866)

You can create a world file from this .tif file

Now, you can take the file **germtm.tif** and create a world file using the Image Command Tool.

Start the Image Command Tool

1. From the **Tools** menu of ERDAS IMAGINE, select **Image Command Tool**.

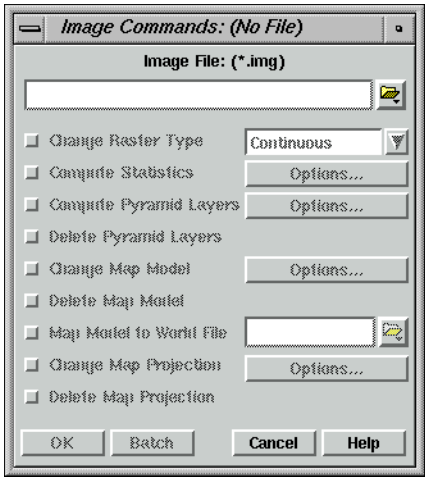


Image Commands: (No File)
Image File: (*.img)
Change Raster Type
Continuous
Compute Statistics
Options...
Compute Pyramid Layers
Options...
Delete Pyramid Layers
Change Map Model
Options...
Delete Map Model
Map Model to World File
Change Map Projection
Options...
Delete Map Projection
OK
Batch
Cancel
Help

2. In the Image Commands dialog, click the Open icon.

The Image File dialog opens.

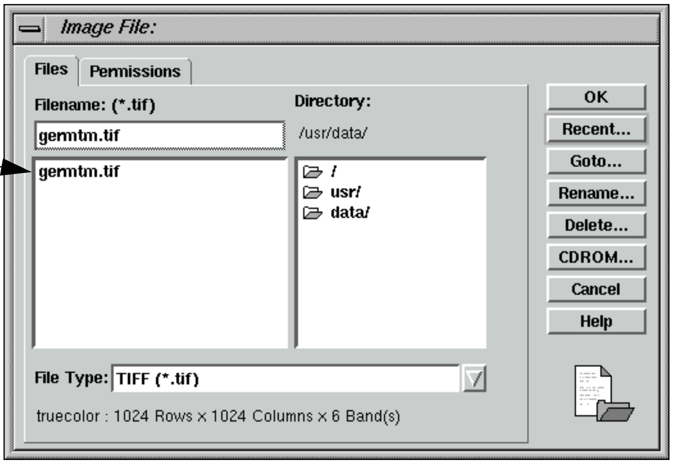


Image File:
Files
Permissions
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
/
usr/
data/

Filename: (*.tif) germtm.tif

Directory: /usr/data/

germtm.tif

File Type: TIFF (*.tif)

truecolor : 1024 Rows × 1024 Columns × 6 Band(s)

Select the file germtm.tif

3. Navigate to the directory in which you saved the file, and select the file **germtm.tif**.

4. Click **OK** in the Image File dialog to transfer the information to the Image Commands dialog.

   The Image Commands dialog updates accordingly, and the **Map Model to World** option is activated.

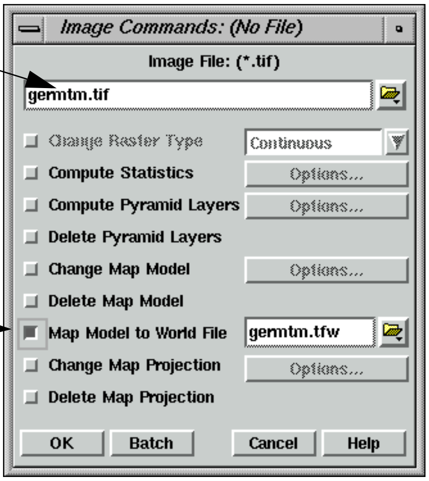


Image Commands: (No File)
Options...
Options...
Options...
Options...
OK
Batch
Cancel
Help

Image File: (*.tif) germtm.tif

☐ Change Raster Type: Continuous
☐ Compute Statistics
☐ Compute Pyramid Layers
☐ Delete Pyramid Layers
☐ Change Map Model
☐ Delete Map Model
☒ Map Model to World File: germtm.tfw
☐ Change Map Projection
☐ Delete Map Projection

The Image File displays in this window

Once you click Map Model to World, the field updates automatically

5. Click the checkbox next to **Map Model to World**.

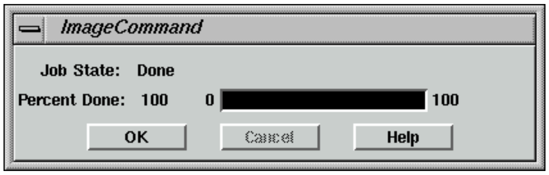
   By default, the Image Commands tool assigns the same name to the output file, but with the .tfw extension. In this case, the file name is **germtm.tfw**. The world file is saved in the same directory as the image you create it from.

6. If you wish, click the Open icon, and navigate to a different directory in which you want to save the world file. If not, proceed to step 9.

The File Selector dialog opens.

7. Type the name **germtm.tfw** in the **Filename** window, then press Return on your keyboard.
8. Click **OK** in the File Selector dialog.
9. Click **OK** in the Image Commands dialog to start the process.

   A Job Status dialog opens, which tracks the progress of the function.



Job State: Done

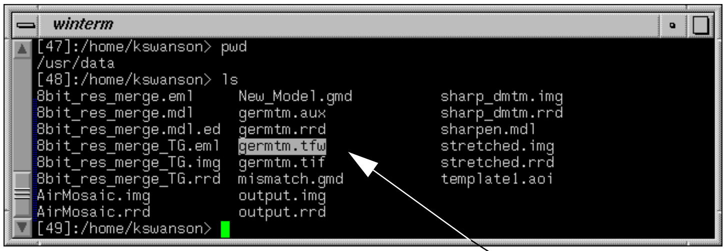
Percent Done: 100

10. When the job is complete, click **OK** in the Job Status dialog.

You can set a preference in the **User Interface & Session Category** *of the Preference Editor,* **Keep Job Status Box***, which allows the Job Status box to close automatically upon completion of a job.*

Check for .tfw file

1. Using a UNIX shell or the Microsoft Explorer, navigate to the directory in which you created **germtm.tif**, and generated **germtm.tfw**.
2. Note the presence of the file in that directory.



```
[47]:/home/kswanson> pwd
/usr/data
[48]:/home/kswanson> ls
8bit_res_merge.eml     New_Model.gmd          sharp_dmtm.img
8bit_res_merge.mdl     germtm.aux             sharp_dmtm.rrd
8bit_res_merge.mdl.ed  germtm.rrd             sharpen.mdl
8bit_res_merge_TG.eml  germtm.tfw             stretched.img
8bit_res_merge_TG.img  germtm.tif             stretched.rrd
8bit_res_merge_TG.rrd  mismatch.gmd           template1.aoi
AirMosaic.img          output.img
AirMosaic.rrd          output.rrd
[49]:/home/kswanson>
```

The .tfw file resides in the same directory as the .tif file it was generated from

Now, you can use the .tfw file to supply georeferencing information to other software packages, such as ArcView. Georeferencing information includes coordinate information.

CHAPTER EIGHT

Import/Export

Introduction

The ERDAS IMAGINE Import function allows you to import a wide variety of data types for use in ERDAS IMAGINE. The Export function lets you convert image (ERDAS IMAGINE .img file format) files into one of several data formats.

In this tour guide, you can learn how to:

- import **SPOT** data from CD
- import **ERDAS 7.x GIS** and **Generic Binary Data** files
- export an image file in the **ERDAS 7.x LAN** format
- create a TIFF file
- view raw data values using the **Data View** option
- view image data information using the **ImageInfo** utility

Approximate completion time for this tour guide varies depending upon the data you are importing or exporting.

Import a SPOT Scene

Before you can import data from a peripheral device, such as a tape drive or a CD-ROM drive, you must configure the device in ERDAS IMAGINE.

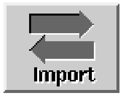
Refer to the current ERDAS IMAGINE Installation Guide for configuration instructions.

This section takes you through the steps to import SPOT data. Since each individual has different types of SPOT data, these steps are only an example.

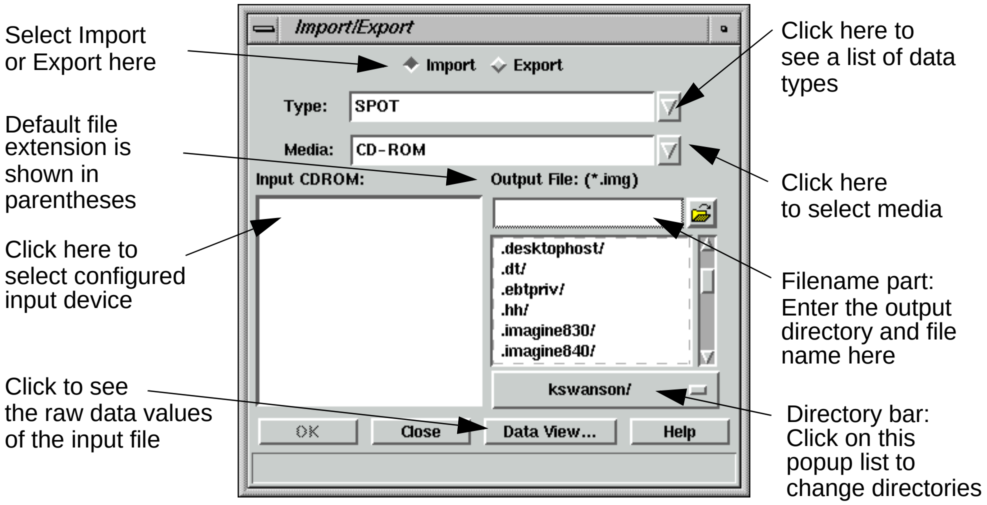
1. If your Session Log is not already open, select **Session | Session Log** from the ERDAS IMAGINE menu bar.

   The Session Log displays real-time messages about what is happening throughout the import process. Following these messages closely helps you understand what is happening.

2. Click on the Import icon on the ERDAS IMAGINE icon panel.



The Import/Export dialog opens:



Type: SPOT

Media: CD-ROM

Select Import or Export here

Default file extension is shown in parentheses

Click here to select configured input device

Click to see the raw data values of the input file

Click here to see a list of data types

Click here to select media

Filename part: Enter the output directory and file name here

Directory bar: Click on this popup list to change directories

The **Import** button is enabled when the dialog first opens, so that all of the prompts that display in the dialog are for importing data. When you click on the **Export** button, the prompts change to options for exporting data.

3. Click on the **Type** popup list to select **SPOT** from the list of available importers.
4. Confirm that the **Media** defaults to **CD ROM**.

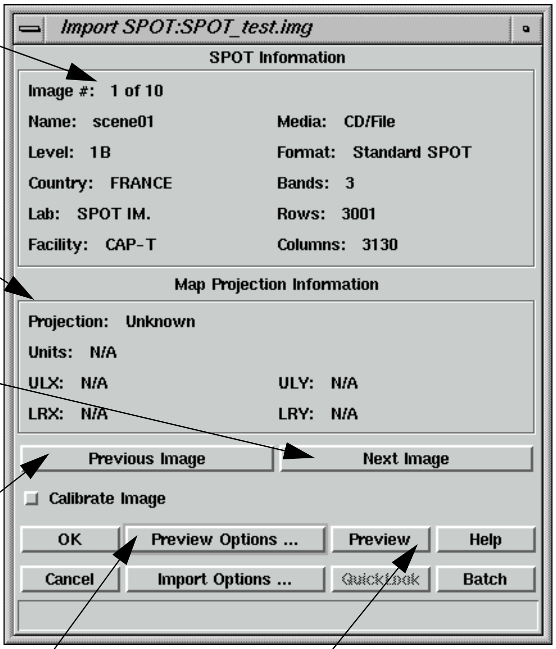
   Displayed under **Input CDROM** is a list of configured CD-ROM drives.

5. Click to select the device from which you want to import data.

If it is necessary to configure a new device, see the current ERDAS IMAGINE Installation Guide.

6. Under **Output File**, enter a name for the output file in the directory of your choice. You can click on the directory bar popup list or enter the directory name in the filename part to select a different directory.
7. Click **OK** in the Import/Export dialog.
8. Watch the Session Log to see a report of the activities of the import process.

   The Import SPOT dialog opens:



Import SPOT:SPOT_test.img

SPOT Information

Image #: 1 of 10

Name: scene01

Media: CD/File

Level: 1B

Format: Standard SPOT

Country: FRANCE

Bands: 3

Lab: SPOT IM.

Rows: 3001

Facility: CAP-T

Columns: 3130

Map Projection Information

Projection: Unknown

Units: N/A

ULX: N/A

ULY: N/A

LRX: N/A

LRY: N/A

Previous Image
Next Image
☐ Calibrate Image
OK
Preview Options ...
Preview
Help
Cancel
Import Options ...
QuickLook
Batch

Shows information about the selected image

Shows map projection information, if available

Click to go to the next image on CD, if a multi-image CD

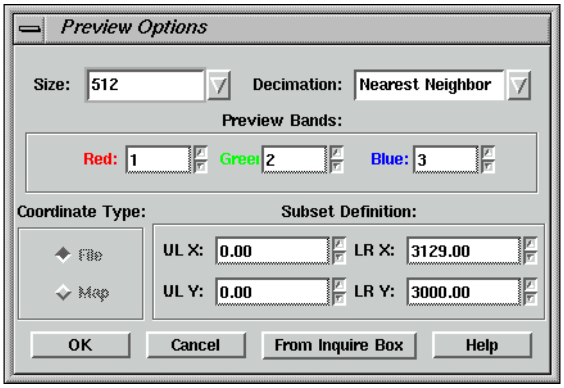
Click to go to the previous image on CD, if a multi-image CD

Click here to define options for the preview image

Click here to see a preview image before importing the data

Check Preview Options

1. Click on **Preview Options** and the following dialog opens:



Preview Options

Size: 512

Decimation: Nearest Neighbor

Preview Bands:

Red: 1

Green: 2

Blue: 3

Coordinate Type:

◆ File

◇ Map

Subset Definition:

UL X: 0.00

LR X: 3129.00

UL Y: 0.00

LR Y: 3000.00

OK
Cancel
From Inquire Box
Help

Preview Options Dialog

The Preview Options allow you to import a decimated (i.e., reduced) version of the image file. You can view the image before importing to be sure that this is the image you want, or to define a subset area of the image to import.

The Preview Options dialog is the same for all importers. For example, if you were importing a Landsat TM scene and you wanted to preview the image, the Preview Options dialog would be identical to the one on page 161.

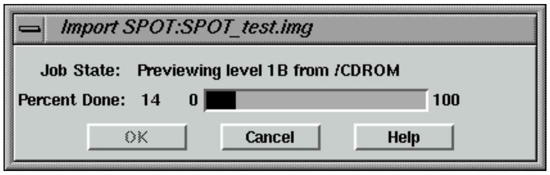
In the Preview Options dialog, you can set:

- **Size** of the displayed image—up to 512 × 512 or 1024 × 1024.
- **Decimation**—**Nearest Neighbor** (provides a faster preview image, although it may appear blocky) or **Average Pixel** (averages groups of pixels, which provides a smooth image but may take longer, especially with large images).
- **Preview Bands**—the bands that are assigned the red, green, and blue colors in the previewed image (this option may not be applicable to some data).
- **Coordinate Type**—the type of coordinates used to define the subset. If there is no map information in the source image, only **File** coordinates are available.
- **Subset Definition**—define a subset area for previewing or importing.

2. Click **OK** in the Preview Options dialog.

3. Click **Preview** in the Import SPOT dialog.

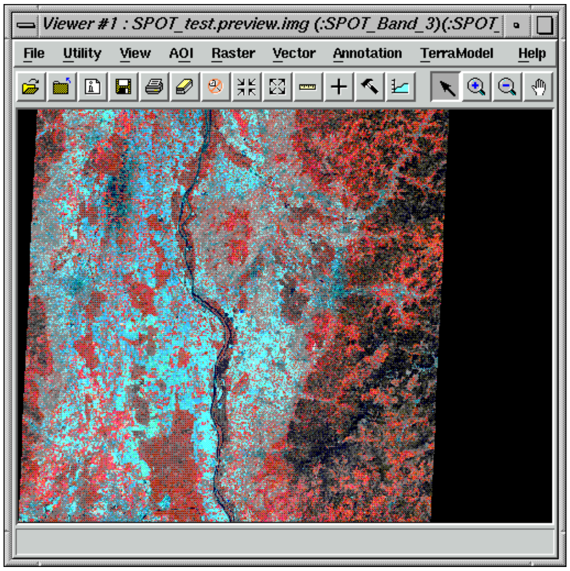
A Job Status dialog displays, indicating the progress of the function.


Import SPOT:SPOT_test.img

Job State: Previewing level 1B from /CDROM

Percent Done: 14

When the Job Status bar shows 100 (indicating that the job is 100% done), the dialog automatically closes. A Viewer window similar to the following opens and displays the preview image:

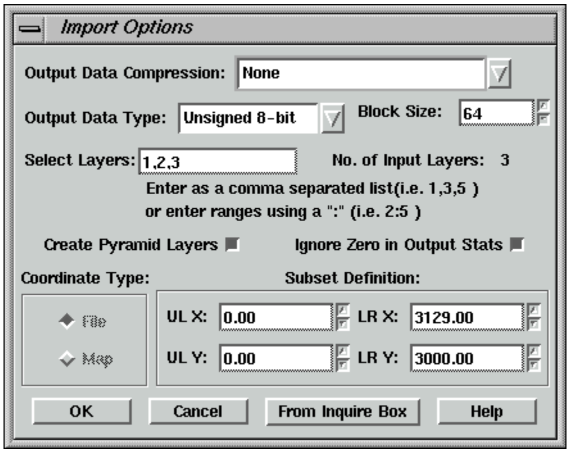

Viewer #1 : SPOT_test.preview.img (:SPOT_Band_3)(:SPOT_
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Preview images have a **.preview.img** *file extension. Because the preview image is decimated, it is not suitable for image processing.*

Check Import Options

1. In the Import SPOT dialog, click on **Import Options**.

   The Import Options dialog opens. This dialog is similar to the Preview Options dialog.



Import Options

Output Data Compression: None

Output Data Type: Unsigned 8-bit

Block Size: 64

Select Layers: 1,2,3

No. of Input Layers: 3

Enter as a comma separated list(i.e. 1,3,5)
or enter ranges using a ":" (i.e. 2:5)

Create Pyramid Layers ☐

Ignore Zero in Output Stats ☐

Coordinate Type:

● File
○ Map

Subset Definition:

UL X: 0.00

LR X: 3129.00

UL Y: 0.00

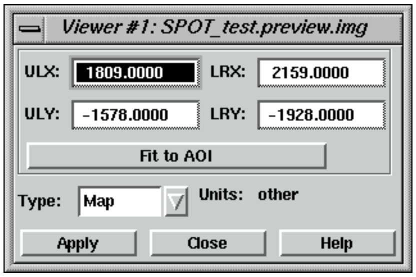
LR Y: 3000.00

OK
Cancel
From Inquire Box
Help

NOTE: Like the Preview Options dialog, the Import Options dialog is the same for every type of importer.

2. Select **Utility | Inquire Box** from the **SPOT_test.preview.img** Viewer menu bar.

The Inquire Box Coordinates dialog opens and a rectangular box (the Inquire Box) is displayed in the center of the Viewer.



ULX: 1809.0000 LRX: 2159.0000

ULY: -1578.0000 LRY: -1928.0000

Type: Map Units: other

Subset an Area

You can select a subset area to be imported by moving and resizing the Inquire Box in the Viewer:

- to resize the Inquire Box, hold and drag on the sides or corners of the box
- to move the Inquire Box, hold and drag in the center of the box

The coordinates update in the Inquire Box dialog as you move and/or resize the Inquire Box.

3. When you have selected the subset area to import, click on **From Inquire Box** in the Import Options dialog.

The coordinates for the subset area display in the Import Options dialog.

Sometimes, when map coordinate information is not available in the source image, there are apparent differences between the Inquire Box coordinates and those transferred to the Import Options dialog. However, the imported image subset matches the area bounded by the inquire box.

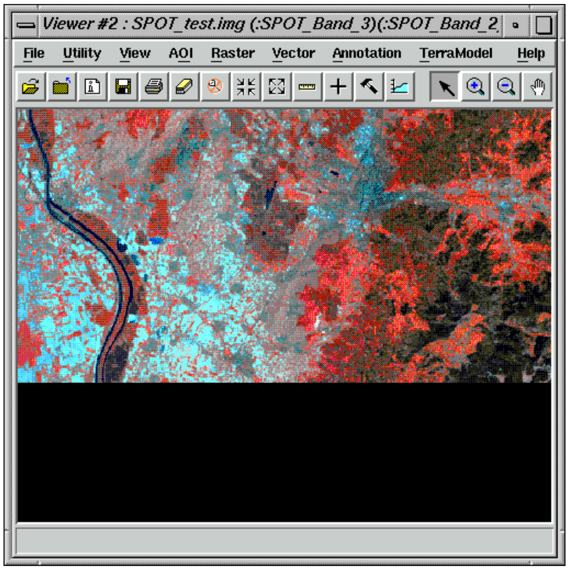
4. Click **OK** in the Import Options dialog.

5. Click **OK** in the Import SPOT dialog.

A Job Status dialog displays, indicating the progress of the import process:

Depending on your eml **Preferences**, when the Job Status bar shows 100 (indicating that the job is 100% done), you must either click **OK** to close the dialog or the dialog closes automatically.

6. Open a Viewer window and display the output file.


Viewer #2 : SPOT_test.img (:SPOT_Band_3)(:SPOT_Band_2
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

See "Chapter 3: Raster and Vector Data Sources" *of the* ERDAS Field Guide *for more information on importing data.*

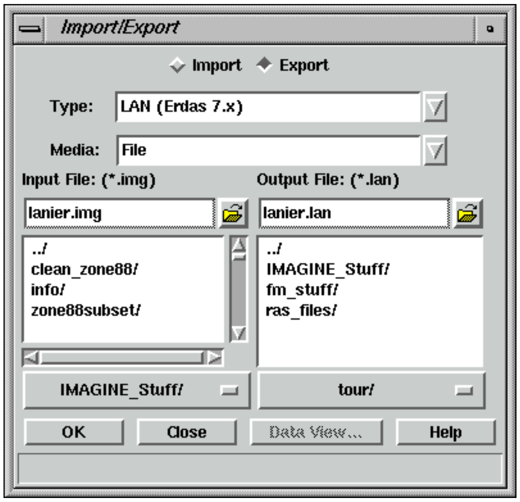
Export LAN Data

In this section, you export one of the image example files to an ERDAS Version 7.x LAN file.

Choose Export Options

ERDAS IMAGINE should be running and the Import/Export dialog should be open.

1. In the Import/Export dialog, click on **Export**.
2. Click on the **Type** popup list to see a list of available exporters
3. Click to select **LAN (Erdas 7.x)** from the **Type** popup list.
4. Confirm that the **Media** lists **File**.
5. Enter **lanier.img** under **Input File**.
6. In the Import/Export dialog, ERDAS IMAGINE automatically enters **lanier.lan** as the output file. You may change this name if you want.


Import/Export
Import
Export
../
clean_zone88/
info/
zone88subset/
../
IMAGINE_Stuff/
fm_stuff/
ras_files/
IMAGINE_Stuff/
tour/
OK
Close
Data View...
Help

○ Import ● Export

Type: LAN (Erdas 7.x)

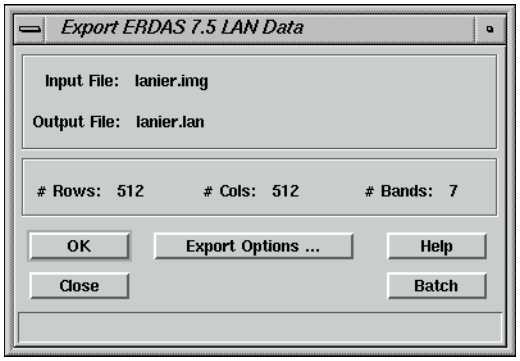
Media: File

Input File: (*.img) lanier.img

Output File: (*.lan) lanier.lan

7. Click **OK** in the Import/Export dialog.

The Export ERDAS 7.5 LAN Data dialog opens:


Export ERDAS 7.5 LAN Data
OK
Export Options ...
Help
Close
Batch

Input File: lanier.img

Output File: lanier.lan

Rows: 512 # Cols: 512 # Bands: 7

8. Click **OK** in the Export ERDAS 7.5 LAN Data dialog.

A Job Status dialog displays.

The file **lanier.lan** now resides in the selected directory.

Creating LAN Data

ERDAS IMAGINE also creates two other files when it exports the LAN data:

- **lanier.pro**—contains map projection information
- **lanier.sta**—contains file statistics

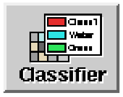
These file formats are used in ERDAS 7.5 software. In ERDAS IMAGINE, map projection and file statistics are included in the image file format. You may delete the files **lanier.pro** and **lanier.sta** if desired, since they are not used in this exercise.

Create .tif Files

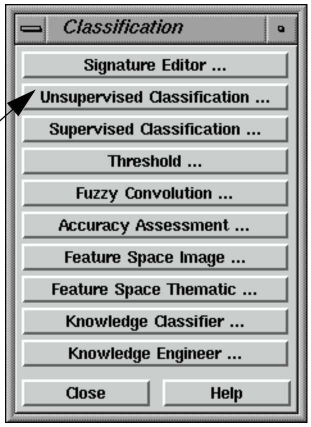
In this section, run an ERDAS IMAGINE process, producing a .tif file instead of an image file.

NOTE: Since the input image is georeferenced, the output TIFF image has geotiff tags. If you wish to produce a TIFF World file, select the Write option in the TIFF image file category of the Preference Editor.

1. Click on the Classifier icon from the ERDAS IMAGINE icon panel.


Class 1
Water
Grass
Classifier

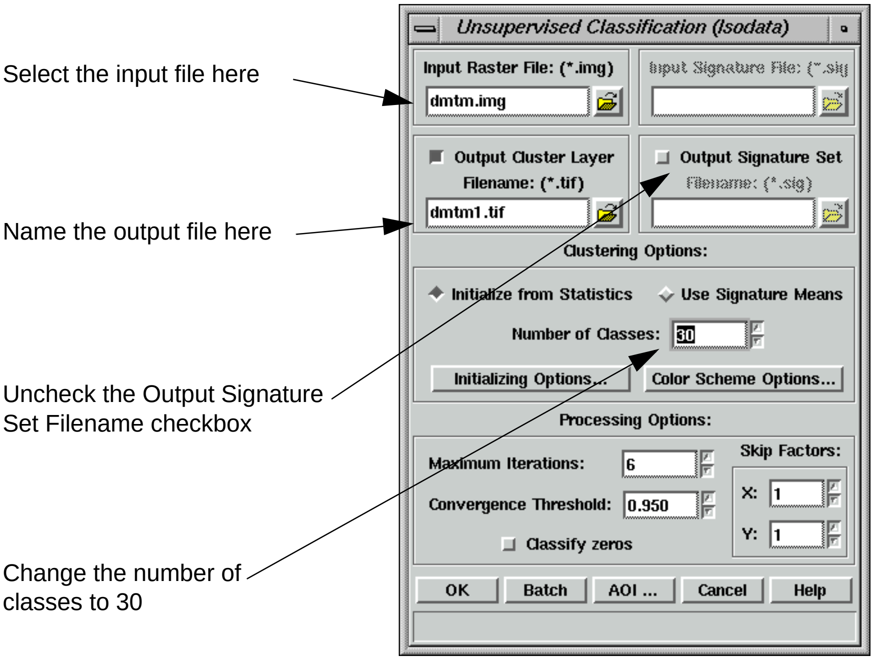
The **Classification** menu opens.


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Close
Help

Click here to start the Unsupervised Classification

2. Click on **Unsupervised Classification** from the **Classification** menu.

The Unsupervised Classification (Isodata) dialog appears.



Unsupervised Classification (Isodata)

Input Raster File: (*.img) dmtm.img

Input Signature File: (*.sig)

☒ Output Cluster Layer Filename: (*.tif) dmtm1.tif

☐ Output Signature Set Filename: (*.sig)

Clustering Options:

● Initialize from Statistics ○ Use Signature Means

Number of Classes: 30

Processing Options:

Maximum Iterations: 6

Convergence Threshold: 0.950

☐ Classify zeros

Skip Factors: X: 1 Y: 1

Select the input file here

Name the output file here

Uncheck the Output Signature Set Filename checkbox

Change the number of classes to 30

3. Click the Open icon by **Input Raster File.**

The Input Raster File dialog opens.

4. In the Input Raster File dialog, click **dmtm.img**, then click **OK**.

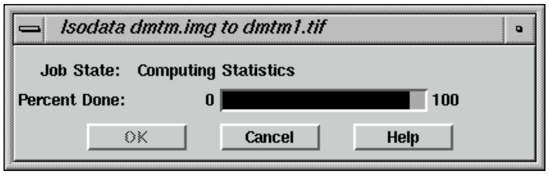
The raster file, **dmtm.img**, *is located in the <IMAGINE_HOME>/examples directory.*

5. Click the Open icon next to **Output Cluster Layer Filename.**

The Filename dialog opens.

6. Navigate to a directory where you have write permission.
7. Click the **File Type** dropdown list, and select **TIFF (*.tif)**.
8. Type in **dmtm1.tif** in the **Filename** field, and then click **OK**.
9. Change the **Number of Classes** to **30**. This tells the classifier how many classes to create.
10. Deselect the **Output Signature Set Filename** checkbox.
11. Click **OK** at the bottom of the Unsupervised Classification dialog.

A job status dialog displays the progress of the classification of the file.


Isodata dmtm.img to dmtm1.tif
OK
Cancel
Help

Job State: Computing Statistics

Percent Done: 0 100

The resulting classification is created in the .tif format.

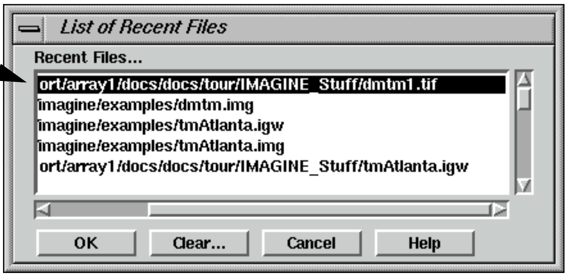
Check the Classification

1. Click the Viewer icon to open a Viewer.

2. From the Viewer tool bar, click the Open icon.

3. In the Select Layer To Add dialog, click the **Recent** button.

4. Select the file **dmtm1.tif** from the top of the list.


List of Recent Files
OK
Clear...
Cancel
Help

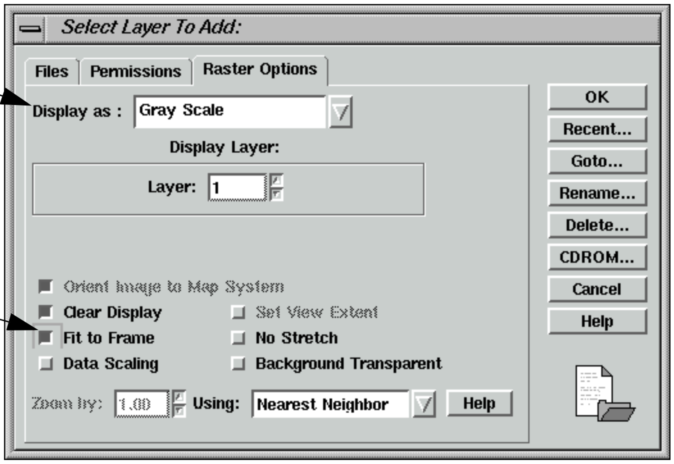
Recent Files...

ort/array1/docs/docs/tour/IMAGINE_Stuff/dmtm1.tif
imagine/examples/dmtm.img
imagine/examples/tmAtlanta.igw
imagine/examples/tmAtlanta.img
ort/array1/docs/docs/tour/IMAGINE_Stuff/tmAtlanta.igw

Select dmtm1.tif from the top of the Recent Files list

5. Click **OK** in the List of Recent Files dialog.

6. Click the **Raster Options** tab of the Select Layer to Add dialog.



Select Gray Scale

Select Fit to Frame

Select Layer To Add:

Display as : Gray Scale

Display Layer:

Layer: 1

☑ Orient Image to Map System

☑ Clear Display

☐ Set View Extent

☑ Fit to Frame

☐ No Stretch

☐ Data Scaling

☐ Background Transparent

Zoom by: 1.00

Using: Nearest Neighbor

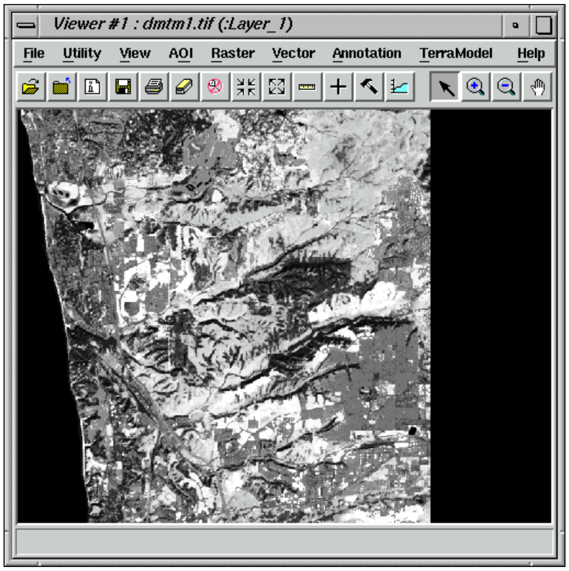
Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
Help

7. Click the **Display as** dropdown list and select **Gray Scale**.

8. Click the **Fit to Frame** checkbox.

9. Click **OK** in the Select Layer To Add dialog.

   The file **dmtm1.tif** displays in the Viewer. The various shades of gray denote different classes of land cover.

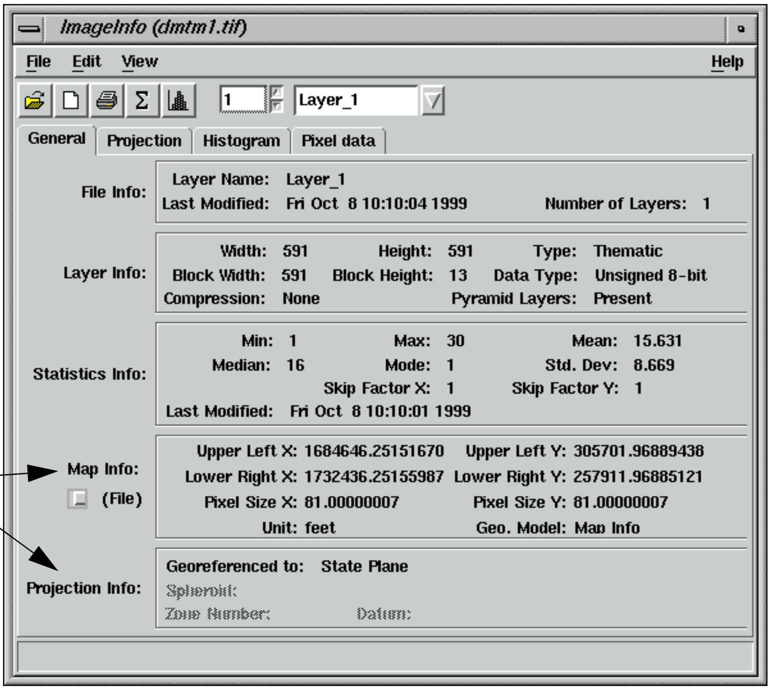


Viewer #1 : dmtm1.tif (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Check Map Information

1. Click the ImageInfo Icon on the Viewer tool bar.

The ImageInfo dialog for **dmtm1.tif** opens.



File Info:
Layer Name: Layer_1
Last Modified: Fri Oct 8 10:10:04 1999
Number of Layers: 1

Layer Info:
Width: 591
Height: 591
Type: Thematic
Block Width: 591
Block Height: 13
Data Type: Unsigned 8-bit
Compression: None
Pyramid Layers: Present

Statistics Info:
Min: 1
Max: 30
Mean: 15.631
Median: 16
Mode: 1
Std. Dev: 8.669
Skip Factor X: 1
Skip Factor Y: 1
Last Modified: Fri Oct 8 10:10:01 1999

Map Info:
☐ (File)
Upper Left X: 1684646.25151670
Upper Left Y: 305701.96889438
Lower Right X: 1732436.25155987
Lower Right Y: 257911.96885121
Pixel Size X: 81.00000007
Pixel Size Y: 81.00000007
Unit: feet
Geo. Model: Map Info

Projection Info:
Georeferenced to: State Plane
Spheroid:
Zone Number:
Datum:

Note map and projection information

Note the information in the **Map Info** section and that the **Projection Info** section shows that the map is georeferenced to **State Plane**.

2. When you are finished, click **File | Close** in the ImageInfo dialog.

3. Click **File | Close** in the Viewer containing **dmtm1.tif**.

Import Generic Binary Data

In this section, you import the **lanier.lan** file that you just exported. This file could be imported using the ERDAS 7.x LAN import function. However, you are using this file to learn how the Generic Binary Data dialog works.

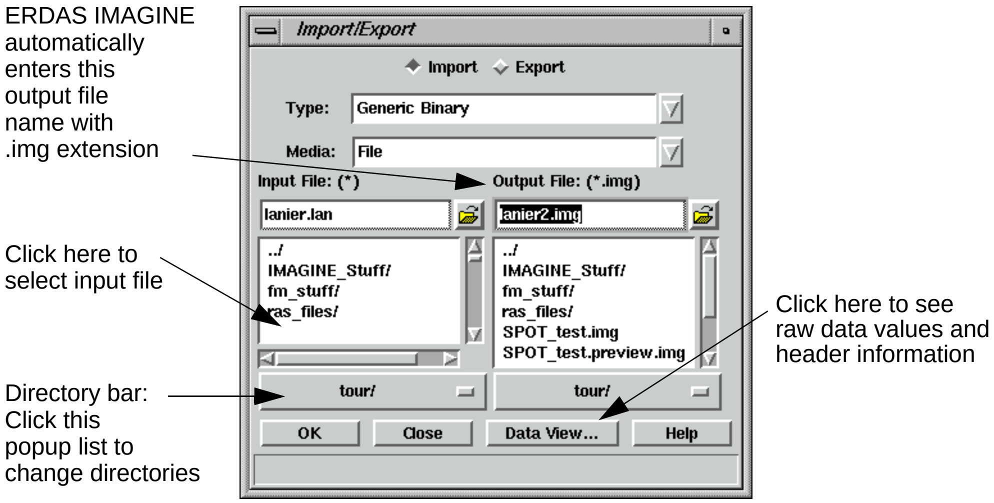
This function is designed for importing data types for which there are currently no specific ERDAS IMAGINE importers.

1. In the Import/Export dialog, click on **Import**.

2. Click on the **Type** popup list and select **Generic Binary**.

3. Select **File** from the **Media** popup list.

4. Under **Input File**, select **lanier.lan**. Change directories if needed by clicking on the directory bar.



Type: Generic Binary

Media: File

Input File: (*) lanier.lan

Output File: (*.img) lanier2.img

ERDAS IMAGINE automatically enters this output file name with .img extension

Click here to select input file

Directory bar: Click this popup list to change directories

Click here to see raw data values and header information

5. Under **Output File**, ERDAS IMAGINE automatically enters **lanier.img** as the file name. If you try to save the file in the <IMAGINE_HOME>/examples directory, an Attention message warns you that **lanier.img** already exists.

6. If the Attention dialog displays, click **No**.

   This prevents the original **lanier.img** file from being overwritten.

7. In the Import/Export dialog, enter **lanier2.img** as the **Output File** in the directory of your choice.

8. In the Import/Export dialog, click **OK**.

   The Import Generic Binary Data dialog opens.

   This dialog requires you make many inputs that define the data structure for ERDAS IMAGINE. In some cases, you can get the information for these inputs from the header data of the file (using the **Data View** option from the Import/Export dialog), but you may have to rely on prior knowledge or assumption for the information.

   You must enter the correct inputs for the data you are importing. The defaults for this dialog are not acceptable for importing.

Data View

1. Click on the **Data View** button in the Import/Export dialog.

   The DataView dialog opens. The data, as initially displayed, may not appear to be very useful. The following steps result in a display from which you can extract the information needed to complete the generic import.

   Data View is also available from the **Utility** *menu on the ERDAS IMAGINE menu bar.*

LAN File Format

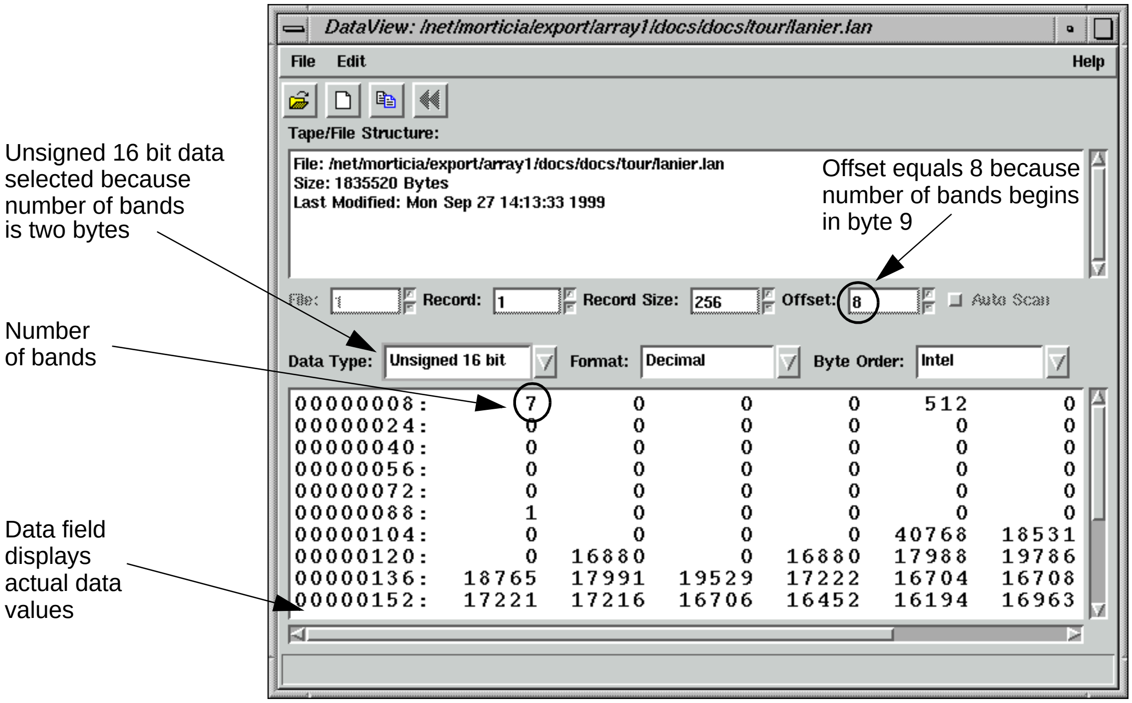
The LAN format has the following characteristics:

- the first 128 bytes of the file are header data
- the number of bands is stored as a 16-bit integer in bytes 9 and 10
- the number of columns is stored as a 32-bit integer in bytes 17-20
- the number of rows is stored as a 32-bit integer in rows 21-24

With this knowledge, you can look at the data values in these bytes to obtain the information needed to import the file as generic binary data.

2. In the DataView dialog, click on **Data Type** and select **Unsigned 16 bit**.
3. In the DataView dialog, click on **Byte Order** and select **Intel**.
4. Set the **Offset** to **8** and check that the **Format** is **Decimal**.

   The DataView dialog is updated to look like the following:


DataView: /net/morticia/export/array1/docs/docs/tour/lanier.lan
File
Edit
Help
Tape/File Structure:
Auto Scan

```
File: /net/morticia/export/array1/docs/docs/tour/lanier.lan
Size: 1835520 Bytes
Last Modified: Mon Sep 27 14:13:33 1999
```

File: 1 Record: 1 Record Size: 256 Offset: 8

Data Type: Unsigned 16 bit Format: Decimal Byte Order: Intel

```
00000008:      7      0      0      0    512      0
00000024:      0      0      0      0      0      0
00000040:      0      0      0      0      0      0
00000056:      0      0      0      0      0      0
00000072:      0      0      0      0      0      0
00000088:      1      0      0      0      0      0
00000104:      0      0      0      0  40768  18531
00000120:      0  16880      0  16880  17988  19786
00000136:  18765  17991  19529  17222  16704  16708
00000152:  17221  17216  16706  16452  16194  16963
```

Unsigned 16 bit data selected because number of bands is two bytes

Number of bands

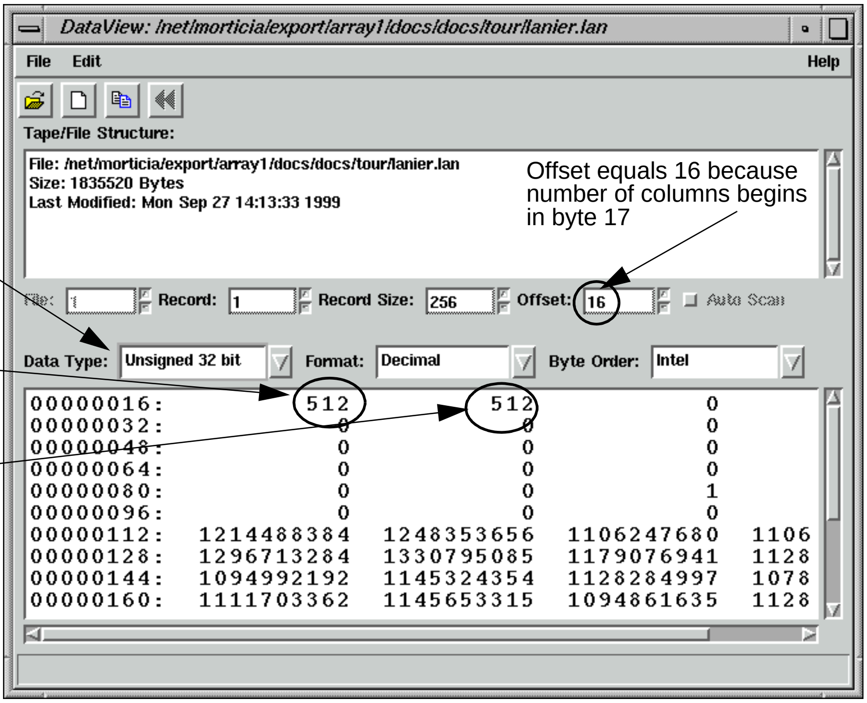
Data field displays actual data values

Offset equals 8 because number of bands begins in byte 9

5. In the DataView dialog, click on **Data Type** and select **Unsigned 32 bit**.
6. In the DataView dialog, click on **Byte Order** and select **Intel**.

7. Set the **Offset** to **16** and check that the **Format** is **Decimal**.

   The DataView dialog is updated.



DataView: /net/morticia/export/array1/docs/docs/tour/lanier.lan
File
Edit
Help
Tape/File Structure:

File: /net/morticia/export/array1/docs/docs/tour/lanier.lan
Size: 1835520 Bytes
Last Modified: Mon Sep 27 14:13:33 1999

File: 1 Record: 1 Record Size: 256 Offset: 16 ☐ Auto Scan

Data Type: Unsigned 32 bit Format: Decimal Byte Order: Intel

```
00000016:         512        512          0
00000032:           0          0          0
00000048:           0          0          0
00000064:           0          0          0
00000080:           0          0          1
00000096:           0          0          0
00000112:  1214488384 1248353656 1106247680 1106
00000128:  1296713284 1330795085 1179076941 1128
00000144:  1094992192 1145324354 1128284997 1078
00000160:  1111703362 1145653315 1094861635 1128
```

Unsigned 32-bit data is selected because the number of columns and rows are four bytes each

Number of columns

Number of rows

Offset equals 16 because number of columns begins in byte 17

Enter Image Dimensions

1. In the Import Generic Binary Data dialog, next to **File Header Bytes**, enter **128**.

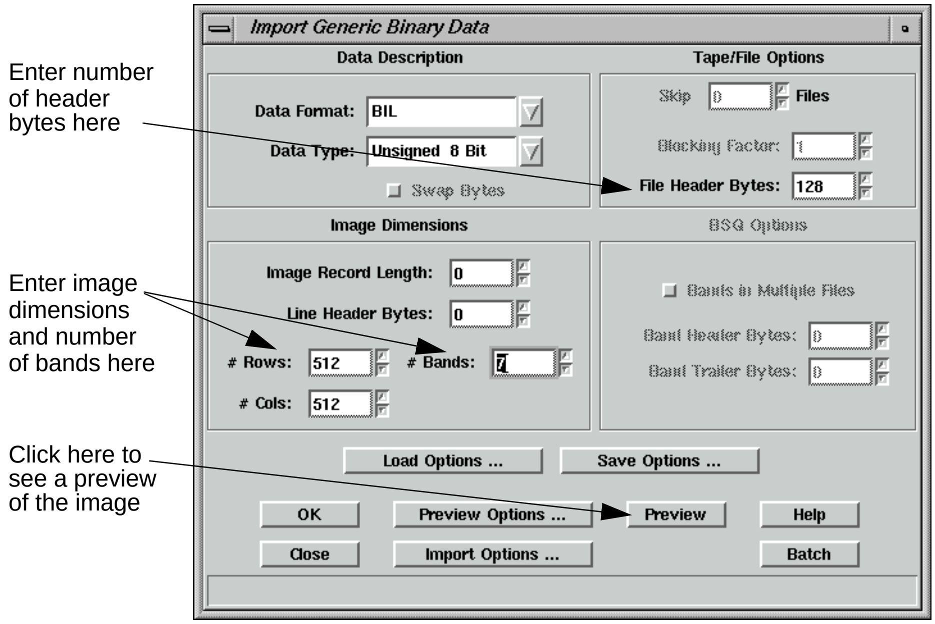
   This is the number of bytes in the header, before the actual data values begin.

2. Under **Image Dimensions**, for **# Rows**, enter **512**.

3. Under **Image Dimensions**, for **# Cols**, enter **512**.

4. Under **Image Dimensions**, for **# Bands**, enter **7**.

   The Import Generic Binary Data dialog is updated.


Enter number of header bytes here

Enter image dimensions and number of bands here

Click here to see a preview of the image

Import Generic Binary Data
Data Description
Tape/File Options
Image Dimensions
BSQ Options
Load Options ...
Save Options ...
OK
Preview Options ...
Preview
Help
Close
Import Options ...
Batch

Data Format: BIL

Data Type: Unsigned 8 Bit

☐ Swap Bytes

Skip: 0 Files

Blocking Factor: 1

File Header Bytes: 128

Image Record Length: 0

Line Header Bytes: 0

Rows: 512

Cols: 512

Bands: 7

☐ Bands in Multiple Files

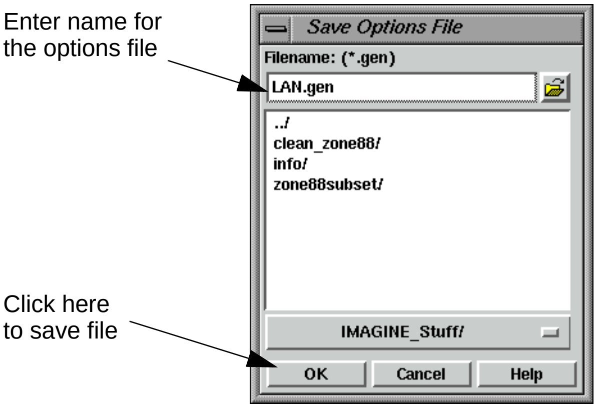
Band Header Bytes: 0

Band Trailer Bytes: 0

Save Options

1. Click on **Save Options**.

   The Save Options File dialog opens.


Enter name for the options file

Click here to save file

Save Options File
../
clean_zone88/
info/
zone88subset/
IMAGINE_Stuff/
OK
Cancel
Help

Filename: (*.gen) LAN.gen

   This dialog allows you to save the inputs you have entered to a .gen file. It can be recalled and loaded into the Import Generic Data dialog whenever you are importing data with this structure. This keeps you from having to find and enter the necessary inputs again.

2. Under **File name**, enter **LAN** and move the cursor out of the text field.

   ERDAS IMAGINE automatically appends the default .gen file extension. Use a descriptive name that helps you to recognize the type of inputs saved in the file.

3. Click **OK** in the Save Options File dialog.

   These inputs are now saved and can be used again, when appropriate.

   To load the .gen file at a later time, click on the **Load Options** *button in the Import Generic Binary Data dialog.*

Preview the Image

1. In the Import Generic Binary Data dialog, click on **Preview**.

   A Job Status dialog displays. When the Job Status bar shows 100 (indicating that the job is 100% done), the dialog closes automatically.

   A Viewer automatically opens and displays the file **lanier2.preview.img**. The .preview file extension is added to indicate that this is a decimated preview file (not suitable for processing).

2. Open another Viewer window and display the file **lanier.img**.

   You see that the two files are similar in appearance. The difference in these files is that **lanier.img** has map projection information and **lanier2.preview.img** does not. This is because when **lanier.img** was exported to a .lan file, the map information was exported to **lanier.pro**, which is a format used in ERDAS 7.5. The map projection data in **lanier.pro** was not imported with the Generic Binary Data importer.

   However, the imported image does contain statistics (similar to the statistics exported to **lanier.sta**), because statistics are recalculated after a raster file is imported.

3. In the ERDAS IMAGINE menu bar, select **Session | Close All Viewers**.

Get Image Information

The ImageInfo utility is a function that gives you information about ERDAS IMAGINE image files. With this function you can access any image file and see the:

- date of last modification
- number of Layers
- layer information: Width, Height, Type, Block Width, Block Height, Data Type, Compression, and Pyramid Layer status for each layer in the file
- calculated statistics
- map coordinates, when available
- map projection information, when available

NOTE: **ImageInfo** *can only be used for image files. Use* **Vector Info** *for vector coverages.*

Edit Image Information

In image files that have write permission, you can edit file information or perform the following operations using the ImageInfo dialog:

- change layer name
- compute statistics
- add, change, or delete map information
- add, change, or delete map projection

- delete calibration
- access the Raster Attribute Editor to change file attributes, such as color, histogram values, etc.

View Image Information

In this section, you use the ImageInfo utility to learn more about the **lanier.img** file.

1. Select **Tools | Image Information** from the ERDAS IMAGINE menu bar.

The ImageInfo dialog opens.

This option is also accessible from **Session | Tools | Image Information** *on the ERDAS IMAGINE menu bar.*

2. Select **File | Open** from the ImageInfo menu bar, or click the Open icon on the tool bar.

The Image Files dialog opens. This dialog allows you to select a file for which information is displayed in the ImageInfo dialog.

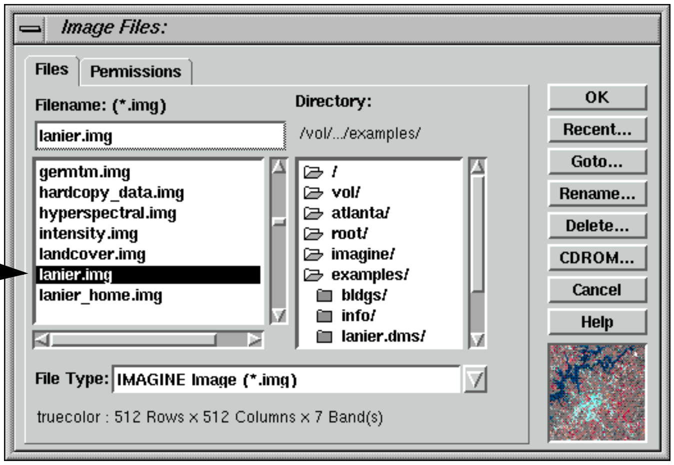


Image Files:
Files
Permissions
germtm.img
hardcopy_data.img
hyperspectral.img
intensity.img
landcover.img
lanier.img
lanier_home.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) lanier.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

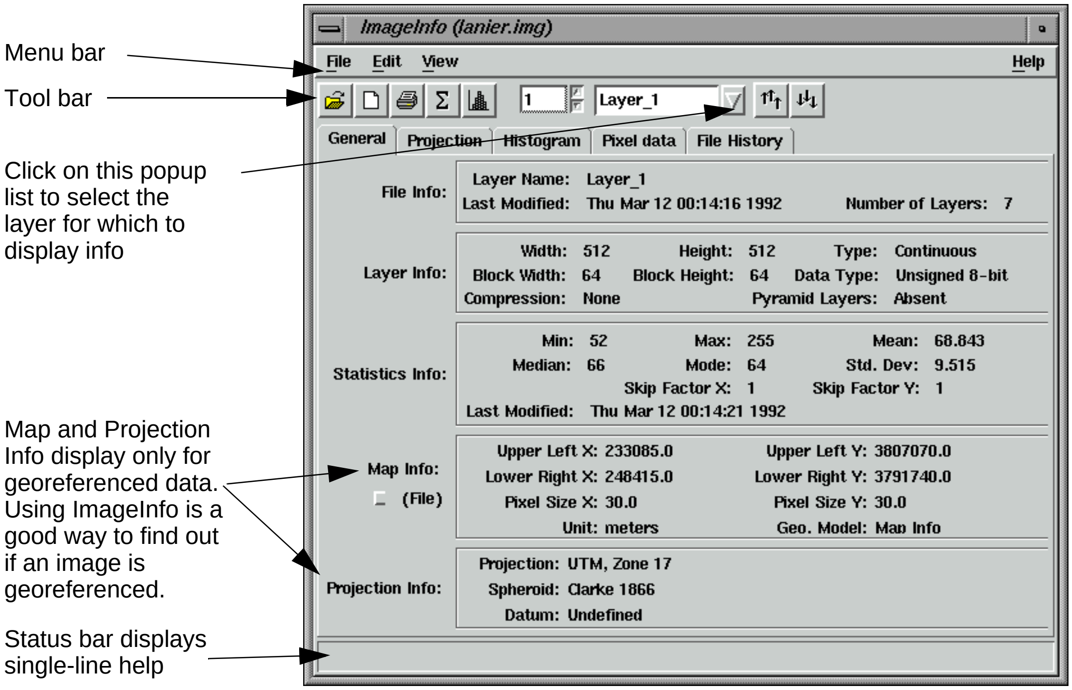
truecolor : 512 Rows x 512 Columns x 7 Band(s)

Select lanier.img

3. Under **Filename**, select **lanier.img**.

4. Click **OK** in the Image Files dialog.

The information for **lanier.img** displays in the ImageInfo dialog.



Menu bar

Tool bar

Click on this popup list to select the layer for which to display info

Map and Projection Info display only for georeferenced data. Using ImageInfo is a good way to find out if an image is georeferenced.

Status bar displays single-line help

File Info: Layer Name: Layer_1 Last Modified: Thu Mar 12 00:14:16 1992 Number of Layers: 7

Layer Info: Width: 512 Height: 512 Type: Continuous Block Width: 64 Block Height: 64 Data Type: Unsigned 8-bit Compression: None Pyramid Layers: Absent

Statistics Info: Min: 52 Max: 255 Mean: 68.843 Median: 66 Mode: 64 Std. Dev: 9.515 Skip Factor X: 1 Skip Factor Y: 1 Last Modified: Thu Mar 12 00:14:21 1992

Map Info: ☐ (File) Upper Left X: 233085.0 Upper Left Y: 3807070.0 Lower Right X: 248415.0 Lower Right Y: 3791740.0 Pixel Size X: 30.0 Pixel Size Y: 30.0 Unit: meters Geo. Model: Map Info

Projection Info: Projection: UTM, Zone 17 Spheroid: Clarke 1866 Datum: Undefined

5. Look over the information listed in the dialog.

The functions in the ImageInfo menu bar in the tool bar are described in the On-Line Help.

Since this is a write-protected file, none of these values can be changed. If you want to practice editing image information with this dialog, you can copy **lanier.img** to the directory of your choice and change the write protections. Then open the writable file in the ImageInfo dialog and click on **Edit** to use the editing functions.

6. Select **File | Close** from the ImageInfo menu bar when you have read the information in the dialog.

CHAPTER NINE

Batch Processing

Introduction

The ERDAS IMAGINE Batch Wizard allows you to process one or more files with one or more commands at any time, from immediately to many years in the future.[1] This is useful if you have a process that requires a long time to run and you want to run it when your system is at minimum utilization (e.g., during the night). It is also useful if you wish to run a repetitive task many times, such as executing the reprojection command to reproject hundreds of images.

Some of the ERDAS IMAGINE functions that can be included in a Batch job are:

- classification
- rectification
- radar processing
- Image Interpreter
- import/export
- reproject

Most of the processing dialogs in ERDAS IMAGINE, such as Importers, Exporters, Image Interpreter functions, Data Preparation functions, and others have a **Batch** button. This button is used to place the command in the Batch queue instead of actually performing the operation. Off-line processing allows unattended operation, enabling you to log off if you wish.

The Batch wizard is distributed as part of the IMAGINE Essentials module, but can be used with functions from other modules. Consequently, some portions of this chapter may not compute successfully unless you also have an IMAGINE Advantage license, because functions from that module are used in the examples.

On systems where off-line processing is not available, the options for processing at a later time are not available.

On Microsoft Windows NT, your account must have administrator privilege to schedule jobs at a later time.

Approximate completion time for this tour guide varies, depending upon the data you select for Batch processing.

---

[1] Under Windows NT, a job cannot be scheduled beyond the end of the current month.

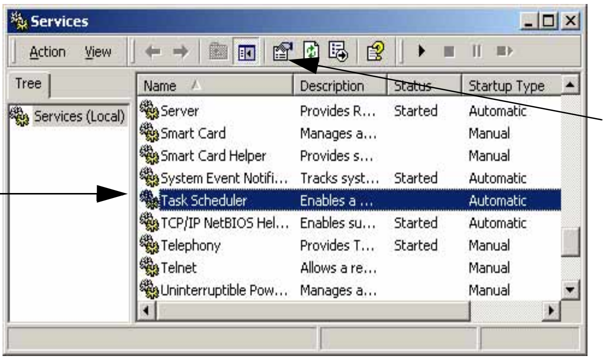
Set up/Start the Task Scheduler on NT and 2000

Before you can use the Batch Wizard to schedule jobs to run at a later time on Windows NT systems, you must set up and run the Scheduler. This requirement means that the account that starts Scheduler must remain logged-in for the scheduled job to run. You must log in as an administrator or as a person with administrator privileges to set up and start the Task Scheduler. Once started, you no longer need to have administrator privileges to submit jobs.

There are two ways to determine if Task Scheduler is installed on your system. If the Explorer shows a folder called “Scheduled Tasks” under “My Computer”, then Task Scheduler is installed. If there is an entry called Task Scheduler under the “Services” control panel, then Task Scheduler is installed. If Task Scheduler is installed, there is an additional panel at the end of the Batch Wizard which collects username and password information required by the Task Scheduler. The “Finish” button will be disabled and the “Next” button will be enabled. Once the information has been entered, it will be remembered for the session and does not have to be entered again. The information is not saved, so when IMAGINE is exited, it will be forgotten and will have to be reentered during the next session. This is a security step because saving it in a file might be a security problem.

1. From the **Start** menu, select **Settings | Control Panel**.
2. On the Control Panel, select **Administrative Tools**, then choose **Services**.

Select Task Scheduler from the Services dialog

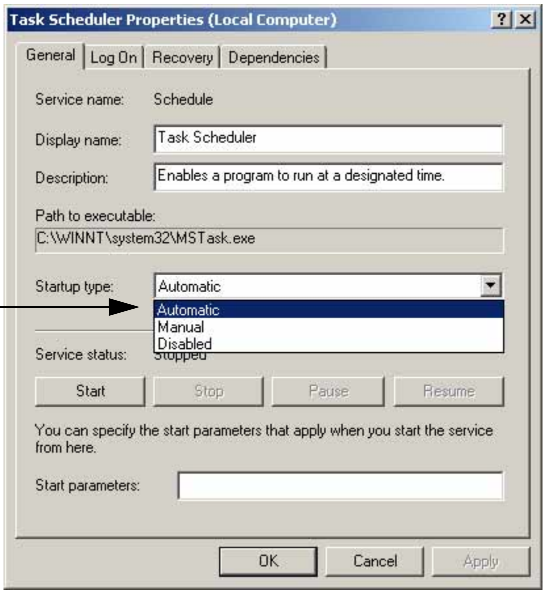


Name	Description	Status	Startup Type
Server	Provides R…	Started	Automatic
Smart Card	Manages a…		Manual
Smart Card Helper	Provides s…		Manual
System Event Notifi…	Tracks syst…	Started	Automatic
Task Scheduler	Enables a …		Automatic
TCP/IP NetBIOS Hel…	Enables su…	Started	Automatic
Telephony	Provides T…	Started	Manual
Telnet	Allows a re…		Manual
Uninterruptible Pow…	Manages a…		Manual

Select Properties from the Services dialog

3. Scroll through the Services window and select **Task Scheduler**, then click the **Properties** button on the Services menu to startup and open the Task Scheduler Properties dialog.

Read the help for the services control panel when using it for the first time. The help button is located on the Services menu bar.



Service name: Schedule

Display name: Task Scheduler

Description: Enables a program to run at a designated time.

Path to executable:
C:\WINNT\system32\MSTask.exe

Startup type: Automatic

- Automatic
- Manual
- Disabled

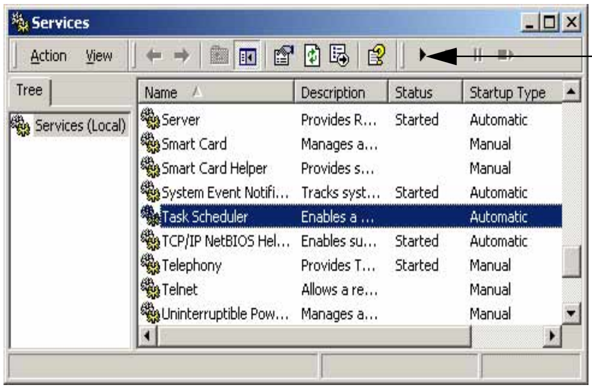
Service status: Stopped

You can specify the start parameters that apply when you start the service from here.

Start parameters:

Select Automatic

4. In the Tax Scheduler Properties dialog, select **Automatic** as the **Startup Type** in the General menu, then click the **This Account** checkbox in the Log On menu. Enter your account name and password, confirm your password, and then click **OK**.



Name	Description	Status	Startup Type
Server	Provides R...	Started	Automatic
Smart Card	Manages a...		Manual
Smart Card Helper	Provides s...		Manual
System Event Notifi...	Tracks syst...	Started	Automatic
Task Scheduler	Enables a ...		Automatic
TCP/IP NetBIOS Hel...	Enables su...	Started	Automatic
Telephony	Provides T...	Started	Manual
Telnet	Allows a re...		Manual
Uninterruptible Pow...	Manages a...		Manual

ClickStartbutton
to start the
Scheduler

5. Click the **Start** button in the Services dialog.

   NOTE: The ability to run Batch at a later time is built using the Windows scheduler, which is part of the **at** *command. Refer to the Windows help to learn more about the use of the* **at** *command.*

   NOTE: (Microsoft Windows NT) If you are submitting jobs for processing later and you are using mapped network drives, then you need to ensure that these maps are in place when the Batch job runs. Typically, a mapped drive is only available to the current user unless the **Reconnect at Logon** *option was checked on the Map Network Drive dialog when the drive was mapped.*

If this option is not checked, then when the Batch process is run by the Scheduler, it is not able to find the drive. This is one of the reasons to make sure that the Schedule service runs as your account name. This ensures that remembered network drive maps are reestablished when the Batch process runs.

Use Batch with a UNIX System

Unlike the Scheduler on NT, you do not have to be an administrator or a person with administrator privileges to set up Batch to run on a UNIX system.

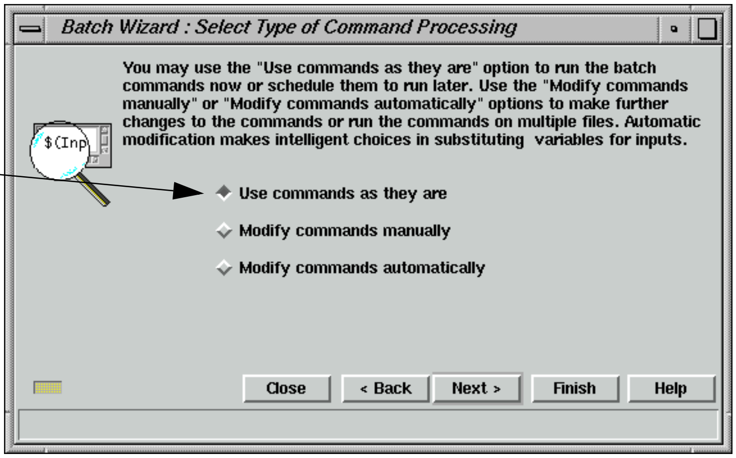
On UNIX, the ability to run Batch at a later time is built using the UNIX **cron** system. See the man pages for more information about **cron** and the **at** command on UNIX.

Execute a Single File/Single Command

This operation is very useful for performing lengthy processes on large files.

1. From the ERDAS IMAGINE icon bar, click the **Interpreter** icon and then select **GIS Analysis | Clump** to open the Clump dialog.
2. In the Clump dialog, select <IMAGINE_HOME>/examples/**lnsoils.img** for the **Input File** name and <your_workspace>/**lnsoils_clump.img** for the **Output File** name.

   Where <your_workspace> is the **Default Output Directory** specified in the **User Interface & Session** category in the Preference Editor, and <IMAGINE_HOME> is the directory where ERDAS IMAGINE is installed.
3. In the Clump dialog, click the **Batch** button. The Batch Wizard opens with the **Select Type of Command Processing** panel displayed. The **Use commands as they are** button is already selected.



Use commands as they are is already selected for you

Batch Wizard : Select Type of Command Processing

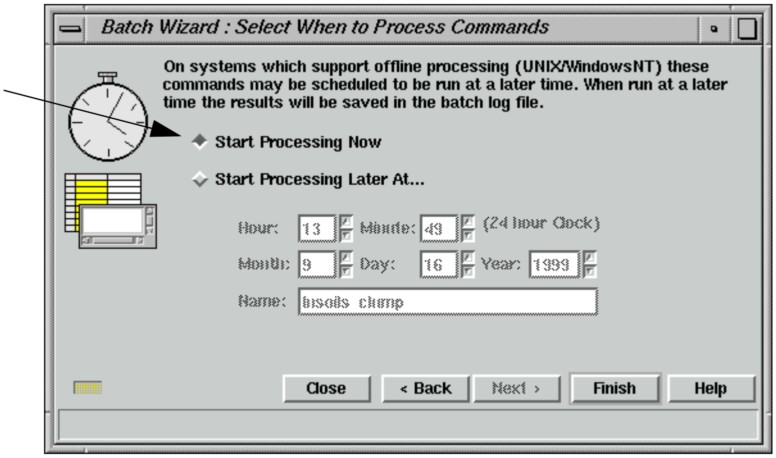
You may use the "Use commands as they are" option to run the batch commands now or schedule them to run later. Use the "Modify commands manually" or "Modify commands automatically" options to make further changes to the commands or run the commands on multiple files. Automatic modification makes intelligent choices in substituting variables for inputs.

◆ Use commands as they are

◇ Modify commands manually

◇ Modify commands automatically

4. Click the **Next >** button. The **Select When to Process Commands** panel displays. The **Start Processing Now** button is already selected. If you wish to process the file immediately, go to step 6.



The Batch job is scheduled to begin now

Batch Wizard : Select When to Process Commands

On systems which support offline processing (UNIX/WindowsNT) these commands may be scheduled to be run at a later time. When run at a later time the results will be saved in the batch log file.

◆ Start Processing Now

◇ Start Processing Later At...

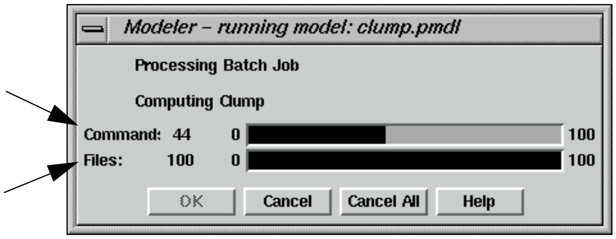
Hour: 13 Minute: 49 (24 hour Clock)

Month: 9 Day: 16 Year: 1999

Name: bsoils clump

5. If you wish to process the file at a later time, click the **Start Processing Later At...** option, and set the time you wish it to begin processing.

6. Click the **Finish** button in the **Select When to Process Commands** panel to begin processing. A double Job Status dialog opens showing the progress of each individual file along with progress of the overall job. When the job is complete, the **OK** button is enabled.



This meter tracks the progress of an individual file

This meter tracks the progress of all the files in the Batch job

Modeler – running model: clump.pmdl

Processing Batch Job

Computing Clump

Command: 44

Files: 100

7. Click **OK** to close the status meter.

The clump operation is complete. If you wish, you can open a Viewer and check the file.

Execute Multiple Files/Single Command—Run Now

The ability to execute a command on multiple files is very useful. In the following example, the Batch Wizard is used to compute statistics for several TIFF images.

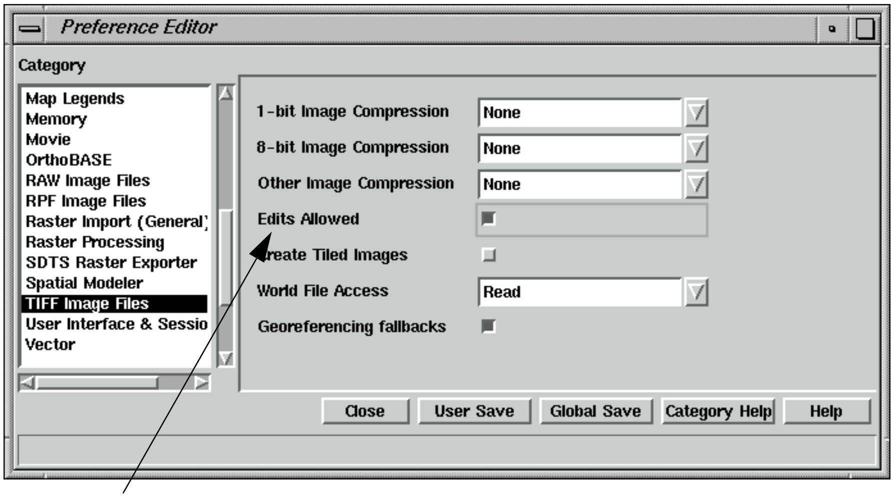
NOTE: You must provide your own TIFF files for this exercise.

See "Chapter 7: Image Commands" *on page 153 for instructions on how to use Image Interpreter utilities to create TIFF images from image files.*

It is important to note that some processes depend upon the physical extents of the imagery, or the number of bands, or data types, or the projection parameters, etc. You must be aware of both the requirements of the processes (commands) as well as the differences between the file types to be processed. In many cases, the data sets provided to multifile processing jobs must share common physical and ephemeral aspects.

Set TIFF Image File Preferences

1. Select **Session | Preferences** from the ERDAS IMAGINE menu bar to open the Preference Editor dialog, and select the **TIFF Image Files** category.


Preference Editor
Category
Map Legends
Memory
Movie
OrthoBASE
RAW Image Files
RPF Image Files
Raster Import (General)
Raster Processing
SDTS Raster Exporter
Spatial Modeler
TIFF Image Files
User Interface & Sessio
Vector
Close
User Save
Global Save
Category Help
Help

1-bit Image Compression: None
8-bit Image Compression: None
Other Image Compression: None
Edits Allowed: ☑
reate Tiled Images: ☐
World File Access: Read
Georeferencing fallbacks: ☑

Make sure that edits are allowed

2. Ensure that the **Edits Allowed** option is enabled (checkbox checked). If you want this to be the default, click **User Save**. Click **Close**.

   Now you are ready to start the Image Command Tool, which provides Batch access to many of the functions that the Image Information tool provides interactively.

Start the Image Command Tool

1. Select **Tools | Image Command Tool** from the ERDAS IMAGINE menu bar to open the Image Commands dialog.
2. Click the File Open icon on the Image Commands dialog. In the File Selector dialog, click the **Files of Type** popup list and select **TIFF**. Select a TIFF file and click **OK**. The options on the Image Commands dialog are now enabled.

   NOTE: If you do not have a TIFF file, one is created in the exercises of "Chapter 7: Image Commands" *on page 153.*

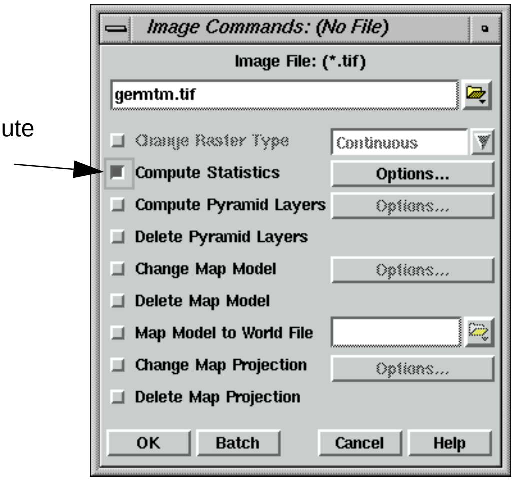


Image File: (*.tif) germtm.tif

☐ Change Raster Type Continuous
☑ Compute Statistics
☐ Compute Pyramid Layers
☐ Delete Pyramid Layers
☐ Change Map Model
☐ Delete Map Model
☐ Map Model to World File
☐ Change Map Projection
☐ Delete Map Projection

Select the Compute Statistics option

3. Check the **Compute Statistics** checkbox on the Image Commands dialog.

Start the Batch Wizard

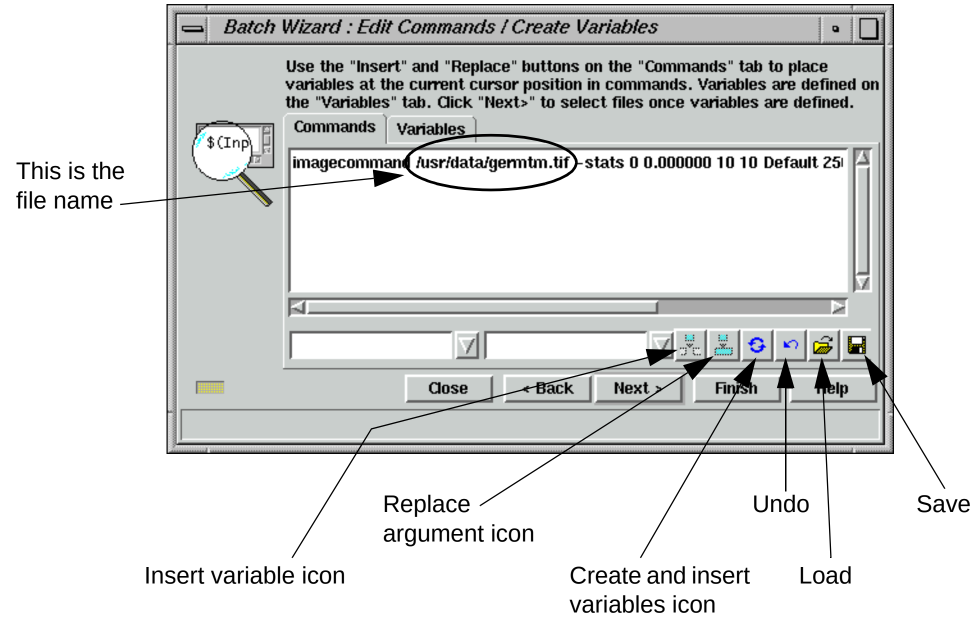
1. Click the **Batch** button in the Image Commands dialog.

   The Batch Wizard starts, and the **Select Type of Command Processing** panel displays. If you click the **Finish** button now, the job runs on the selected file. However, it is more useful to automate the process to run on several files.

2. Select the **Modify commands manually** option in the **Select Type of Command Processing** panel.

3. Click the **Next >** button to display the **Edit Commands/Create Variables** panel.

   Notice that the command, visible in the window, contains the full path of the selected file. In order for the software to be able to make file name substitutions from a list, there must be a variable into which the substitution is made. This is done in the following step.



Batch Wizard : Edit Commands / Create Variables

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

```
imagecommand /usr/data/germtm.tif -stats 0 0.000000 10 10 Default 25
```

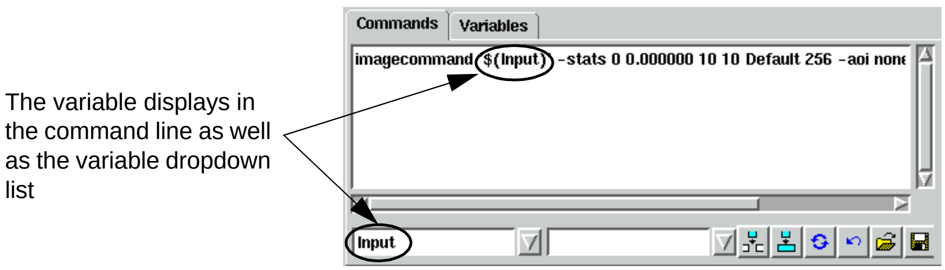
$(Inp
Commands
Variables
Close
< Back
Next >
Finish
Help
This is the file name
Replace argument icon
Insert variable icon
Create and insert variables icon
Undo
Load
Save

4. Click the Create and Insert Variables button to create variables.

This tool examines the command for file names and replaces them with variables. In this case, one variable called **Input** is created and replaces a specific file and path.

For more information about variables, see “Work with Variables” *on page 202.*

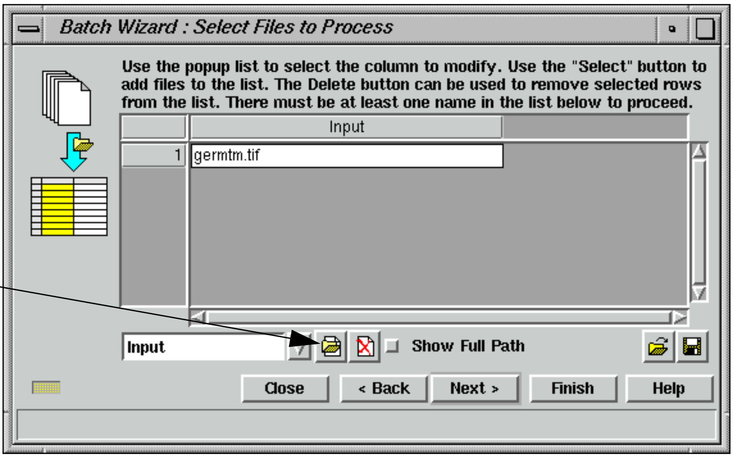
The Batch commands can be saved for future use. See "Save/Load Options" on *page 207.*



```
imagecommand $(Input) -stats 0 0.000000 10 10 Default 256 -aoi non
```

Commands
Variables
Input
The variable displays in the command line as well as the variable dropdown list

5. Click the **Next >** button to display the **Select Files to Process** panel.


Batch Wizard : Select Files to Process
Input
Input
Show Full Path
Close
< Back
Next >
Finish
Help

Use the popup list to select the column to modify. Use the "Select" button to add files to the list. The Delete button can be used to remove selected rows from the list. There must be at least one name in the list below to proceed.

	Input
1	germtm.tif

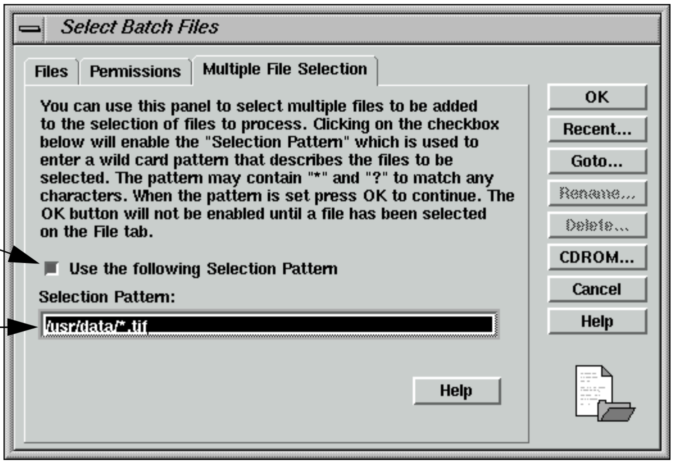
Add files to the Input list by clicking this button

6. Select the Add Files button. This opens the File Selector dialog. Click the **File Type** popup list and select **TIFF**.

NOTE: If you are working on the Windows platform, files can be added to the **Input** *list by dragging and dropping from Explorer.*

7. Select another TIFF file, and then click the **Multiple Files Selection** tab.

8. Click the **Use the following Selection Pattern** checkbox. The default wild card displayed in the **Selection Pattern** field selects all TIFF files in the specified directory. You may modify the wild card as needed.


Select Batch Files
Files
Permissions
Multiple File Selection
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
Help

You can use this panel to select multiple files to be added to the selection of files to process. Clicking on the checkbox below will enable the "Selection Pattern" which is used to enter a wild card pattern that describes the files to be selected. The pattern may contain "*" and "?" to match any characters. When the pattern is set press OK to continue. The OK button will not be enabled until a file has been selected on the File tab.

☑ Use the following Selection Pattern

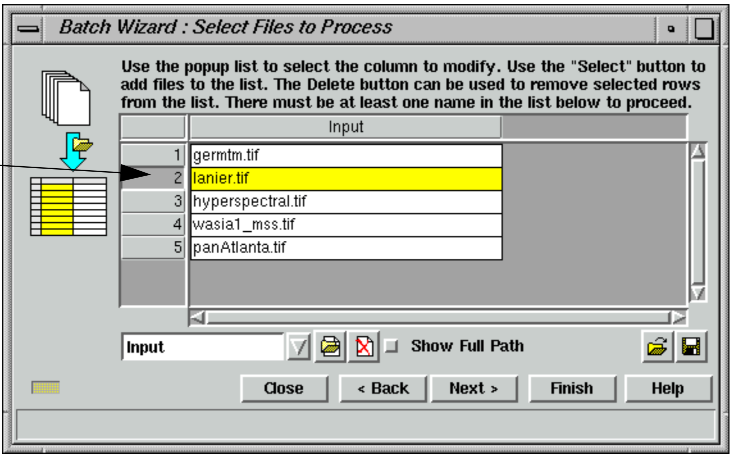
Selection Pattern: /usr/data/*.tif

Click the Use the following Selection Pattern checkbox

The Selection Pattern is based on the file you choose for input

9. Click the **OK** button on in the **Multiple File Selection** tab. The specified TIFF files are added to the list in the **Select Files to Process** panel in the **Input** column.

NOTE: You can repeat step 6. through step 8. to choose TIFF files from other directories.



Use the popup list to select the column to modify. Use the "Select" button to add files to the list. The Delete button can be used to remove selected rows from the list. There must be at least one name in the list below to proceed.

	Input
1	germtm.tif
2	lanier.tif
3	hyperspectral.tif
4	wasia1_mss.tif
5	panAtlanta.tif

Input ☐ Show Full Path

This row is selected, indicated by the yellow highlight

NOTE: Entries in the list may be selected by clicking in the number column as shown above, and then removed from the list with the Delete icon.

10. Click **Finish** to start computing statistics on the listed images now. A double Job Status dialog displays showing the progress of each individual file along with progress of the total job. When the job is complete, the **OK** button is enabled. Click **OK** to close the Job Status dialog.

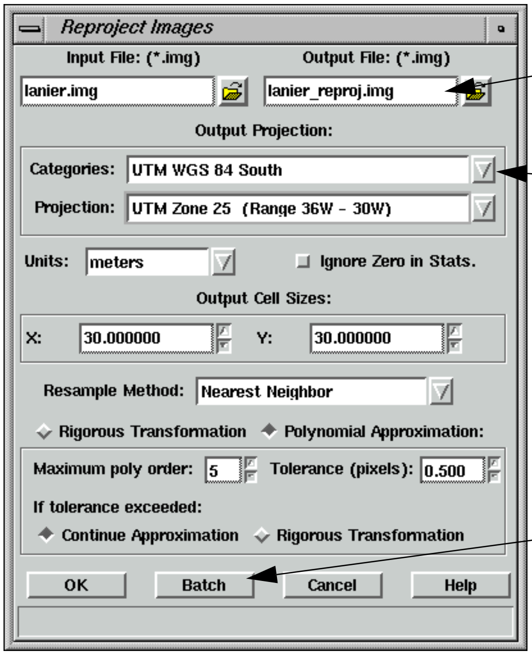
Execute Multiple Files/Single Command—Run Later

In the following example, you are going to perform reprojection on all of the Lake Lanier related files in the ERDAS IMAGINE examples directory.

NOTE: For Windows NT users, you must be an administrator or a person with administrator privileges to run a Batch process at a later time.

1. From the ERDAS IMAGINE icon bar, click the **DataPrep** icon and then select **Reproject Images** to open the Reproject Images dialog. Select **lanier.img** as the **Input File** and **lanier_reproj.img** as the **Output File**. Select **UTM WGS 84 South** from the **Categories** popup list. Select **UTM Zone 25** from the **Projection** popup list.

NOTE: If you are creating projections of your own, do not include a slash (/ or \) in the name. The Batch processor uses the presence of a slash in a name to decide if it is an argument to be converted to a variable.



Input File: (*.img) lanier.img

Output File: (*.img) lanier_reproj.img

Output Projection:

Categories: UTM WGS 84 South

Projection: UTM Zone 25 (Range 36W - 30W)

Units: meters

☐ Ignore Zero in Stats.

Output Cell Sizes:

X: 30.000000 Y: 30.000000

Resample Method: Nearest Neighbor

◇ Rigorous Transformation ◆ Polynomial Approximation:

Maximum poly order: 5 Tolerance (pixels): 0.500

If tolerance exceeded:

◆ Continue Approximation ◇ Rigorous Transformation

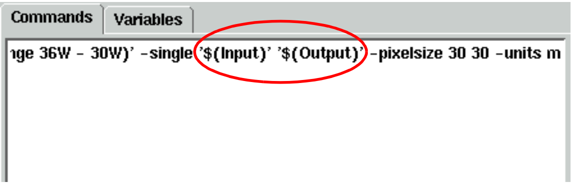
Save the Output File in the directory of your choice

Set the projection here

Click the Batch button

2. Click the **Batch** button. The Batch Wizard starts with the **Select Type of Command Processing** panel. Select the **Modify commands automatically** option.

3. In the Batch Wizard, click the **Next >** button to open the **Edit Commands/Create Variables** panel. Ensure the **Commands** tab is selected. Using the horizontal scroll button, scroll about 2/3 to the left and observe that **Input** and **Output** variables have already been created in the command line.

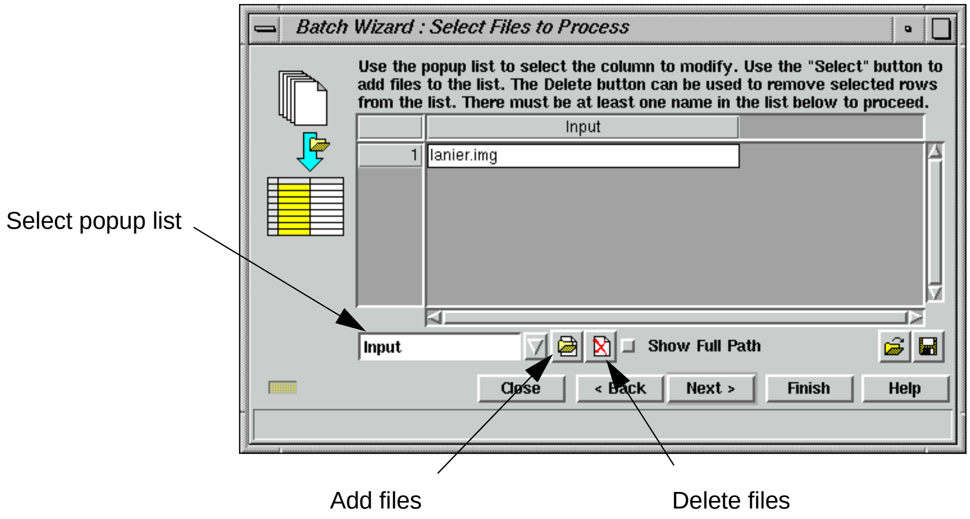


```
ıge 36W - 30W)' -single '$(Input)' '$(Output)' -pixelsize 30 30 -units m
```

For more information about variables, see the “Work with Variables” *section on page 202.*

4. Click the **Next >** button. The **Select Files to Process** panel opens with the selected input file displayed.

   The popup list in the **Select Files to Process** panel determines which variable is populated when the Add Files button is pressed. In this example, there is only an **Input** variable.


Batch Wizard : Select Files to Process
Input
1
lanier.img
Input
Show Full Path
Close
< Back
Next >
Finish
Help
Select popup list
Add files
Delete files

Use the popup list to select the column to modify. Use the "Select" button to add files to the list. The Delete button can be used to remove selected rows from the list. There must be at least one name in the list below to proceed.

By default, there is always an **Input** column in the list, which corresponds to the default **Input** variable.

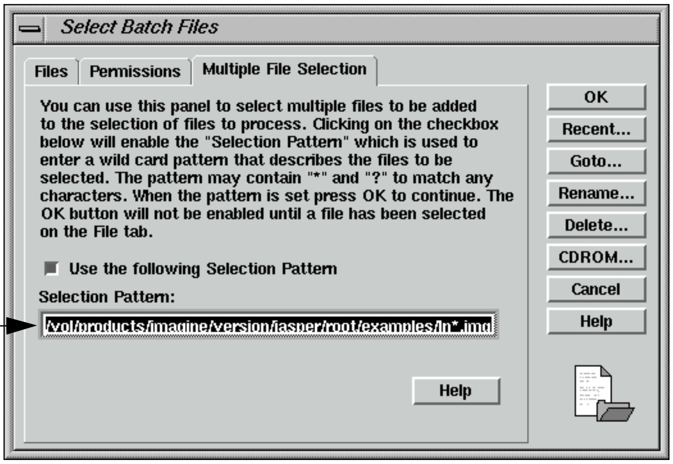
Add Multiple Files

1. Click the Add Files button. The Select Batch Files dialog opens.

2. Select **lnaspect.img** from the Select Batch Files dialog. Click the **Multiple File Selection** tab and click the **Use the following Selection Pattern** checkbox. Edit the **Selection Pattern** field to insert **ln** before the asterisk. It should look like this:

   ```
   <IMAGINE_HOME>/examples/ln*.img
   ```

   Where <IMAGINE_HOME> represents the installation directory of ERDAS IMAGINE on your system.



You can use this panel to select multiple files to be added to the selection of files to process. Clicking on the checkbox below will enable the "Selection Pattern" which is used to enter a wild card pattern that describes the files to be selected. The pattern may contain "*" and "?" to match any characters. When the pattern is set press OK to continue. The OK button will not be enabled until a file has been selected on the File tab.

☑ Use the following Selection Pattern

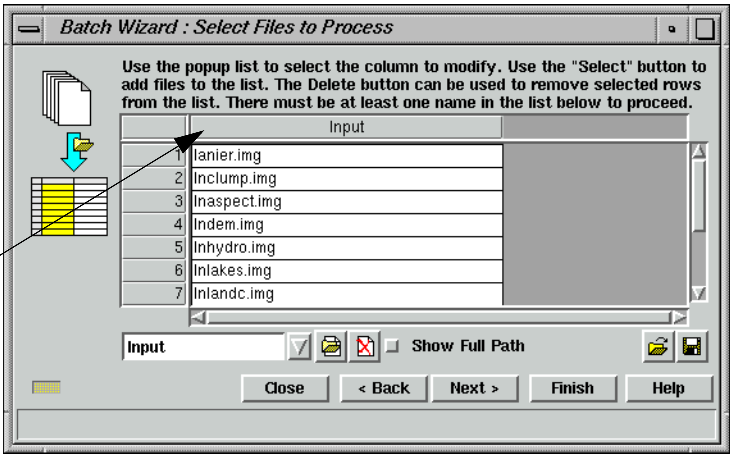
Selection Pattern: /vol/products/imagine/version/jasper/root/examples/ln*.img

The Selection Pattern has been changed to include all files that start with ln

3. Click the **OK** button in the Select Batch Files dialog.

The input files are displayed under the **Input** column. Select and delete any duplicate or unwanted files by clicking in the numbered column next to the file name and then clicking the Delete Files icon. This list of input files may be saved for future use.

See “Additional Information” *on page 207 for information about saving files.*



Use the popup list to select the column to modify. Use the "Select" button to add files to the list. The Delete button can be used to remove selected rows from the list. There must be at least one name in the list below to proceed.

	Input
1	lanier.img
2	lnclump.img
3	lnaspect.img
4	lndem.img
5	lnhydro.img
6	lnlakes.img
7	lnlandc.img

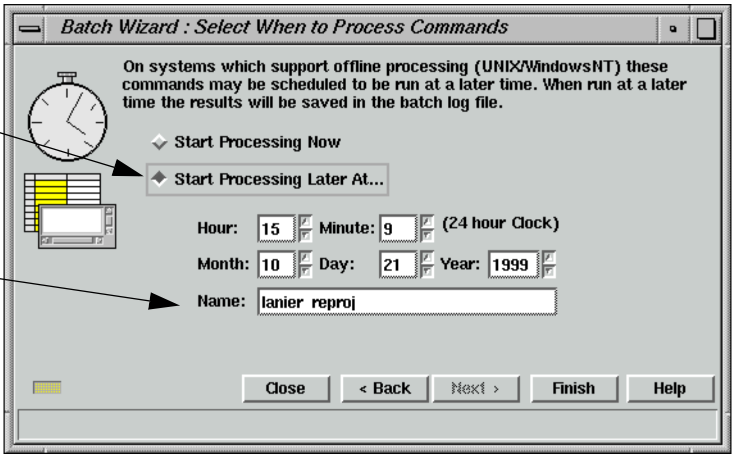
Input ☐ Show Full Path

These are all the files that begin with ln located in the directory identified in the Selection Pattern window

4. Click the **Next >** button to open the **Select When to Process Commands** panel. Click the **Start Processing Later At** checkbox. Use the default time of the next available minute.

This Batch file begins on 21 October 1999 at 3:09 p.m.

When you schedule Batch jobs later, a Name is assigned. Names cannot contain certain characters or spaces. Batch renames the job accordingly.



Batch Wizard : Select When to Process Commands

On systems which support offline processing (UNIX/WindowsNT) these commands may be scheduled to be run at a later time. When run at a later time the results will be saved in the batch log file.

○ Start Processing Now

● Start Processing Later At...

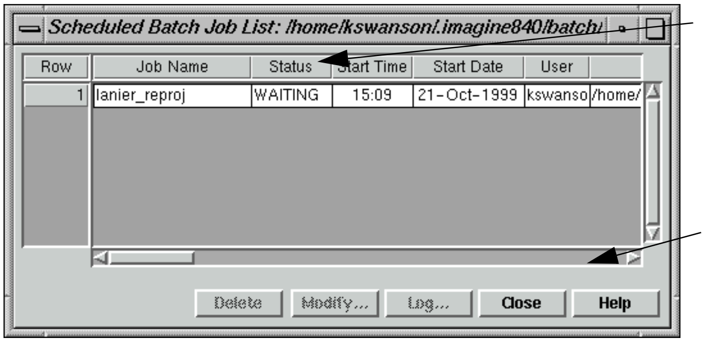
Hour: 15 Minute: 9 (24 hour Clock)

Month: 10 Day: 21 Year: 1999

Name: lanier reproj

The following characters are converted to a **-** *(dash) when present in the* **Name** *field of the* **Select When to Process Commands** *panel:* \, **/**, *****, **?**, **|**, **<**, *and* **>**. *Spaces and tabs are converted to an _ (underscore).*

5. Click the **Finish** button to begin processing. The Scheduled Batch Job List dialog opens, displaying all current Batch jobs.



Scheduled Batch Job List: /home/kswanson/.imagine840/batch/

Row	Job Name	Status	Start Time	Start Date	User	
1	lanier_reproj	WAITING	15:09	21-Oct-1999	kswanso	/home/

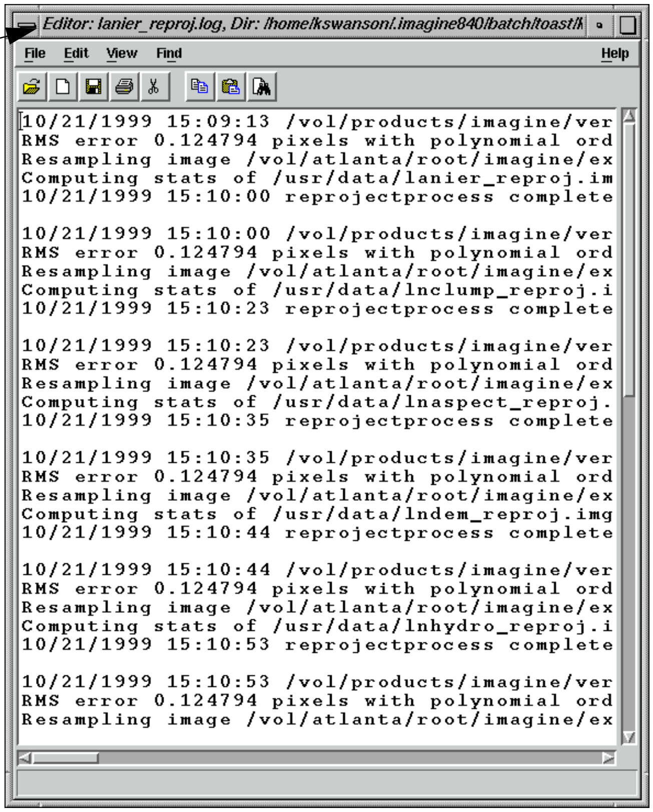
Status starts as WAITING

You can use the scroll bar to see the complete path of the Job File

The Scheduled Batch Job List dialog may be opened any time by selecting **Session | View Off-line Batch Queue** from the ERDAS IMAGINE menu bar.

When the job begins, the **Status** changes from **WAITING** to **ACTIVE**. When the job has finished, the **Status** changes from **ACTIVE** to **DONE**.

When the job is done, the log file can be viewed by first selecting the job in the Scheduled Batch Job list dialog, and then clicking the **Log** button. This is useful for checking to see if your Batch command ran successfully on all of the input files.


Editor: lanier_reproj.log, Dir: /home/kswanson/.imagine840/batch/toast/k
File
Edit
View
Find
Help

```
10/21/1999 15:09:13 /vol/products/imagine/ver
RMS error 0.124794 pixels with polynomial ord
Resampling image /vol/atlanta/root/imagine/ex
Computing stats of /usr/data/lanier_reproj.im
10/21/1999 15:10:00 reprojectprocess complete

10/21/1999 15:10:00 /vol/products/imagine/ver
RMS error 0.124794 pixels with polynomial ord
Resampling image /vol/atlanta/root/imagine/ex
Computing stats of /usr/data/lnclump_reproj.i
10/21/1999 15:10:23 reprojectprocess complete

10/21/1999 15:10:23 /vol/products/imagine/ver
RMS error 0.124794 pixels with polynomial ord
Resampling image /vol/atlanta/root/imagine/ex
Computing stats of /usr/data/lnaspect_reproj.
10/21/1999 15:10:35 reprojectprocess complete

10/21/1999 15:10:35 /vol/products/imagine/ver
RMS error 0.124794 pixels with polynomial ord
Resampling image /vol/atlanta/root/imagine/ex
Computing stats of /usr/data/lndem_reproj.img
10/21/1999 15:10:44 reprojectprocess complete

10/21/1999 15:10:44 /vol/products/imagine/ver
RMS error 0.124794 pixels with polynomial ord
Resampling image /vol/atlanta/root/imagine/ex
Computing stats of /usr/data/lnhydro_reproj.i
10/21/1999 15:10:53 reprojectprocess complete

10/21/1999 15:10:53 /vol/products/imagine/ver
RMS error 0.124794 pixels with polynomial ord
Resampling image /vol/atlanta/root/imagine/ex
```

The log displays in a text editor

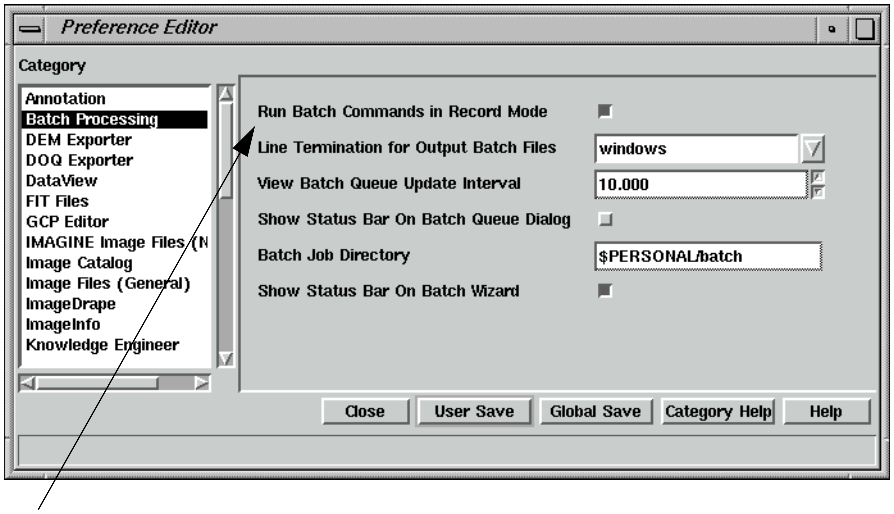
Execute Multiple Files/Multiple Commands

This procedure is most useful for very complex operations on multiple files involving many commands where the output from one command becomes the input to a succeeding command. In the following example, you are going to perform three operations on each of the Lake Lanier related files in the ERDAS IMAGINE examples directory.

It is important to note that some processes depend upon the physical extents of the imagery, or the number of bands, or data types, etc. You must be aware of both the requirements of the processes (commands) as well as the differences between the file types to be processed. In many cases, the data sets provided to multifile processing jobs must share common physical and ephemeral aspects.

Set Up

1. Select **Session | Preferences** from the ERDAS IMAGINE menu bar to open the Preference Editor dialog, and select the **Batch Processing** category.


Preference Editor
Category
Annotation
Batch Processing
DEM Exporter
DOQ Exporter
DataView
FIT Files
GCP Editor
IMAGINE Image Files (N
Image Catalog
Image Files (General)
ImageDrape
ImageInfo
Knowledge Engineer
Close
User Save
Global Save
Category Help
Help

Run Batch Commands in Record Mode ☑
Line Termination for Output Batch Files: windows
View Batch Queue Update Interval: 10.000
Show Status Bar On Batch Queue Dialog ☐
Batch Job Directory: $PERSONAL/batch
Show Status Bar On Batch Wizard ☑

Batch commands should run in record mode

2. Ensure that the **Run Batch Commands in Record Mode** option is enabled (checkbox checked).

   This preference tells ERDAS IMAGINE to run the commands and record them simultaneously. When running in Record Mode, you are typically collecting a series of commands to automate. Since the first command typically produces input for the next command, it is desirable to have the command run as soon as it is placed into the current list of commands.

   Once the command has completed, the resulting file is available to be selected as input to the next command. In this case, the **Create and Insert Variables** button can be used to identify and replace intermediary files with temporary file variables, so that only the final output file is left on disk, and not all of the intermediate steps.

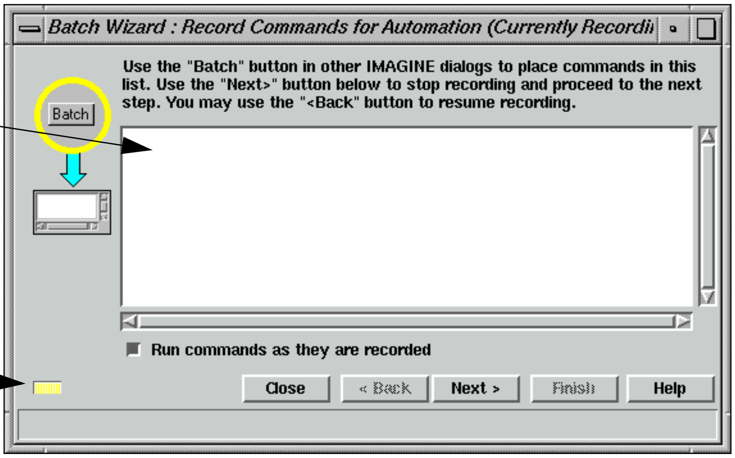
   Optionally, you can select the **Run commands as they are recorded** checkbox on the **Record Commands for Automation** panel. This sets the preference for the current session without changing your general preferences.

3. In the Preference Editor, click the **Image Files (General)** category, then click to deselect the **Compute Pyramid Layers** option. You do not want pyramid layers to be generated for the temporary files. Rather, you can run the Image Command Tool as a Batch process to create pyramid layers for the output files only.

4. Click **User Save** in the Preference Editor, then click **Close**.

5. Select **Session | Start Recording Batch Commands** from the ERDAS IMAGINE menu bar to start the Batch Wizard. The **Record Commands for Automation** panel displays.

   Note that the flashing record light, located in the lower left corner of the Batch Wizard dialog, has been activated.


Batch Wizard : Record Commands for Automation (Currently Recordi
Batch
Close
« Back
Next >
Finish
Help

Use the "Batch" button in other IMAGINE dialogs to place commands in this list. Use the "Next>" button below to stop recording and proceed to the next step. You may use the "<Back" button to resume recording.

☒ Run commands as they are recorded

Batch commands display in this window

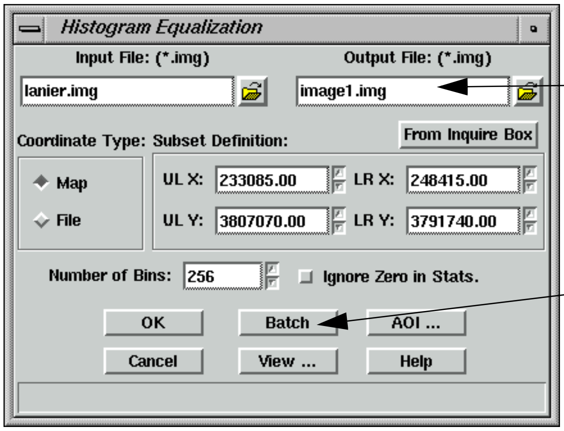
This is the flashing record unit

Run First Command

1. From the ERDAS IMAGINE icon bar, click the **Interpreter** icon and then select **Radiometric Enhancement | Histogram Equalization** to open the Histogram Equalization dialog.

2. Select <IMAGINE_HOME>/examples/**lanier.img** for the **Input File** name and <your_workspace>/**image1.img** for the **Output File** name.

   Where <your_workspace> is the **Default Output Directory** specified in the **User Interface & Session** category in the Preference Editor and <IMAGINE_HOME> is the directory where ERDAS IMAGINE is installed.


Histogram Equalization
From Inquire Box
OK
Batch
AOI ...
Cancel
View ...
Help

Input File: (*.img) lanier.img

Output File: (*.img) image1.img

Coordinate Type: ● Map ○ File

Subset Definition: UL X: 233085.00 LR X: 248415.00 UL Y: 3807070.00 LR Y: 3791740.00

Number of Bins: 256 ☐ Ignore Zero in Stats.

Save the Output File in the directory of your choice

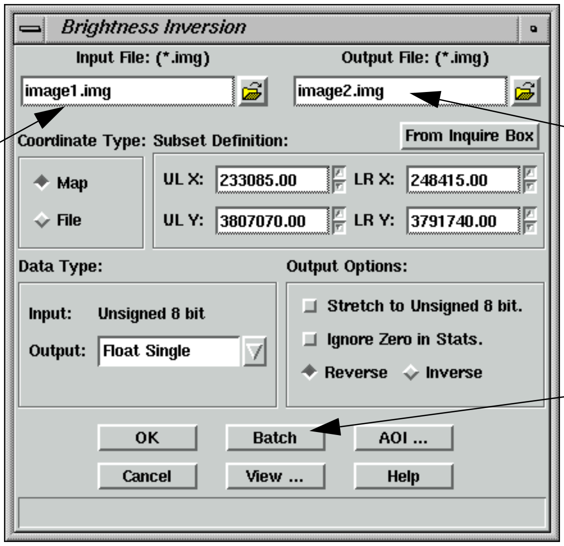
Click the Batch button to add the command to the Batch Wizard

3. Click the **Batch** button in the Histogram Equalization dialog. The command to perform histogram equalization is added to the **Record Commands for Automation** panel. The histogram equalization process starts and a Job Status dialog displays. When the process is done, click **OK**.

Run Next Command

1. To open the Brightness Inversion dialog, select **Brightness Inversion** from the **Radiometric Enhance** menu you opened in the first command.

2. Select <your_workspace>/**image1.img** for the **Input File** name and <your_workspace>/**image2.img** for the **Output File** name. The **Batch** option is now enabled.


Brightness Inversion
Input File: (*.img)
Output File: (*.img)
Coordinate Type:
Subset Definition:
From Inquire Box
Data Type:
Output Options:
OK
Batch
AOI ...
Cancel
View ...
Help

Input File: (*.img) image1.img

Output File: (*.img) image2.img

Coordinate Type: ● Map ○ File

UL X: 233085.00 LR X: 248415.00

UL Y: 3807070.00 LR Y: 3791740.00

Input: Unsigned 8 bit

Output: Float Single

☐ Stretch to Unsigned 8 bit.

☐ Ignore Zero in Stats.

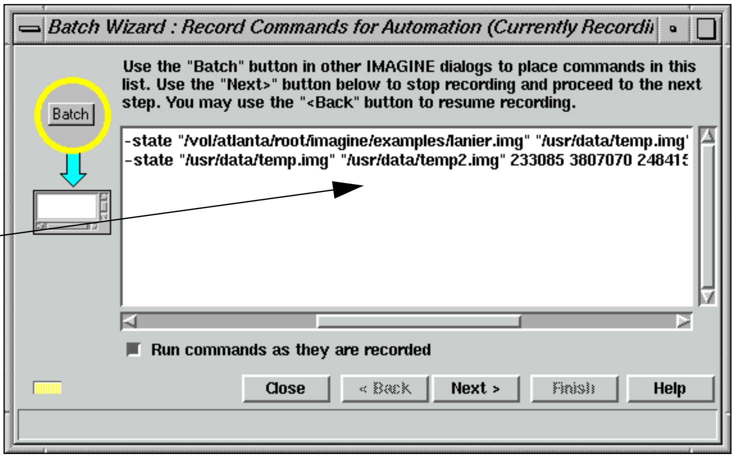
● Reverse ○ Inverse

This is the output file from the first Batch command

The output from the first Batch command is used to create the output for the second Batch command

Click the Batch button to add the second Batch command to the list

3. Click the **Batch** button. The command to perform brightness inversion is added to the **Record Commands for Automation** panel. The histogram brightness inversion starts and a Job Status dialog displays. When the process is done, click **OK**.


Batch Wizard : Record Commands for Automation (Currently Recordi
Batch
Close
< Back
Next >
Finish
Help

Use the "Batch" button in other IMAGINE dialogs to place commands in this list. Use the "Next>" button below to stop recording and proceed to the next step. You may use the "<Back" button to resume recording.

```
-state "/vol/atlanta/root/imagine/examples/lanier.img" "/usr/data/temp.img'
-state "/usr/data/temp.img" "/usr/data/temp2.img" 233085 3807070 24841
```

☒ Run commands as they are recorded

These are the Batch commands you created. Note the exact files involved are specified

Run Another Command

You can also use Batch to create pyramid layers of your data. Now you instruct the Image Command Tool to create pyramid layers for the output files only.

1. From the **Tools** menu of the ERDAS IMAGINE menu bar, select **Image Command Tool**. In the **Image File** window of the Image Commands dialog, select the file you generated in the last example, **image2.img**.

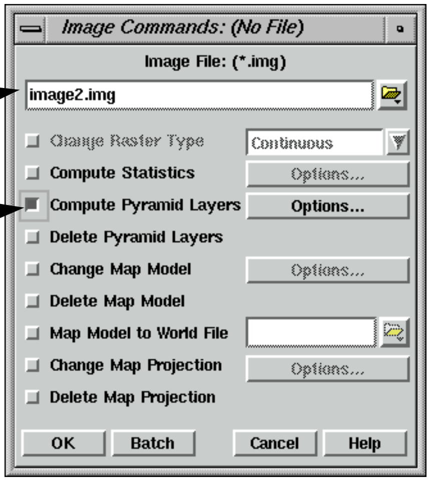


Image Commands: (No File)
Change Raster Type
Compute Statistics
Options...
Options...
Delete Pyramid Layers
Change Map Model
Options...
Delete Map Model
Map Model to World File
Change Map Projection
Options...
Delete Map Projection
OK
Batch
Cancel
Help

Image File: (*.img) image2.img

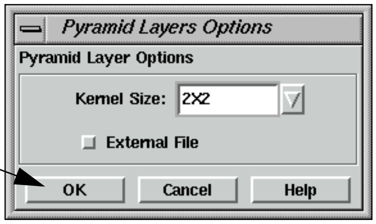
Change Raster Type: Continuous

☒ Compute Pyramid Layers

Select the image output from the last Batch request

Select the Compute Pyramid layers option

2. Click the **Compute Pyramid Layers** checkbox, then click the **Options** button next to **Compute Pyramid Layers** to open the Pyramid Layers Options dialog. Notice the **Kernel Size**, and that the **External File** option is not checked. You want the output file to contain internal pyramid layers rather than having the pyramid layers stored in an external .rrd file. Click **OK** in the Pyramid Layers Options dialog.


Pyramid Layers Options
Pyramid Layer Options
OK
Cancel
Help

Kernel Size: 2X2

☐ External File

Click OK

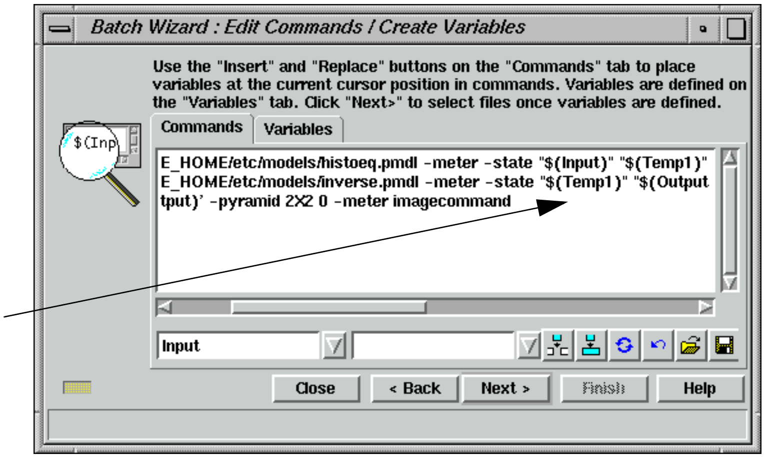
3. Click the **Batch** button in the Image Commands dialog. The command to create pyramid layers is added to the **Record Commands for Automation** panel. The pyramid layer generation starts and a Job Status dialog displays. When the process is done, click **OK**.

   If you examine the contents of the **Record Commands for Automation** panel at this time, you notice that the commands explicitly name the input and output files. The next procedure converts the input and output file names into variables so that these commands may be applied to a larger set of input files.

Create Variables

1. On the **Record Commands for Automation** panel, click the **Next >** button. The **Select Type of Command Processing** panel displays. Select the **Modify commands automatically** option.

2. On the **Select Type of Command Processing** panel, click the **Next >** button. The **Edit Commands/Create Variables** panel displays.


Batch Wizard : Edit Commands / Create Variables
$(Inp
Commands
Variables
Close
< Back
Next >
Finish
Help

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

```
E_HOME/etc/models/histoeq.pmdl -meter -state "$(Input)" "$(Temp1)"
E_HOME/etc/models/inverse.pmdl -meter -state "$(Temp1)" "$(Output
tput)' -pyramid 2X2 0 -meter imagecommand
```

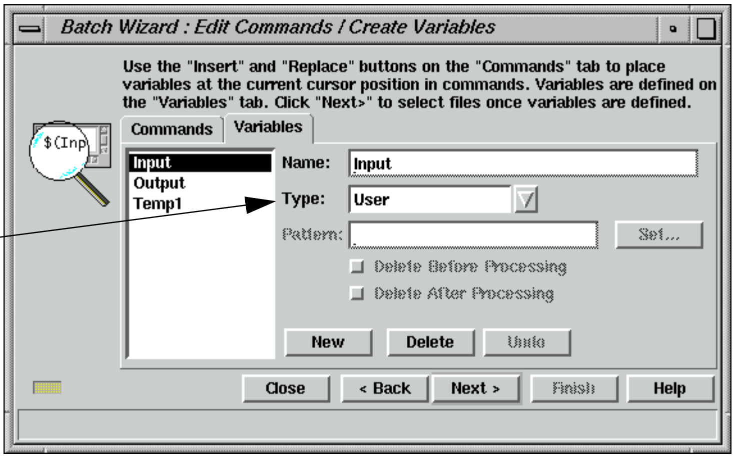
Input

Now, the files have been replaced by variables: Input, Temp1, and Output

Modify Variables

This procedure creates an intermediate file that is used to pass the output of the first process to the input of the second. There is no need to keep this file and use disk space. Also, you might want the output file name to be a little more meaningful than **image2**. The following steps tell you how to modify variables.

1. Click the **Variables** tab to gain access to the variable editing tools.


Batch Wizard : Edit Commands / Create Variables
$(Inp
Commands
Variables
Input
Output
Temp1
Set...
New
Delete
Undo
Close
< Back
Next >
Finish
Help

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Name: Input

Type: User

Pattern:

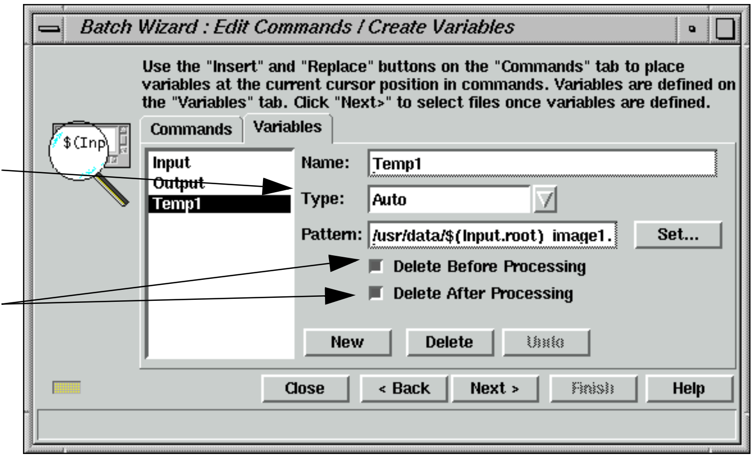
☐ Delete Before Processing

☐ Delete After Processing

The Input variable is defined by you

The **Input** variable is of type **User**. This means that you provide the names of the input files. This is accomplished in subsequent steps using the **Select Files to Process** panel.

2. Select the **Temp1** variable. This represents the intermediate files that are not needed after processing. The **Delete Before Processing** checkbox is automatically selected to remove these files before processing begins. This prevents problems arising from the existence of an old file. The **Delete After Processing** checkbox is also automatically selected to remove these temporary files when processing finishes.



Batch Wizard : Edit Commands / Create Variables

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Name: Temp1

Type: Auto

Pattern: /usr/data/$(Input.root) image1.

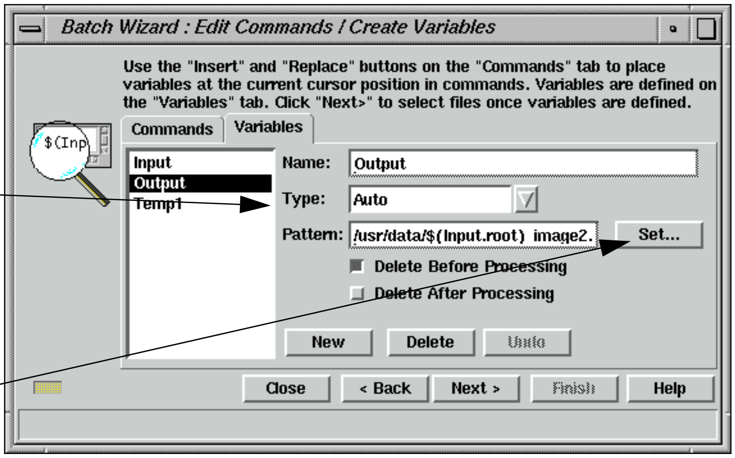
☑ Delete Before Processing

☑ Delete After Processing

The Temp1 variable is automatically generated

Temp1 is deleted both before and after processing

3. Select the **Output** variable. This represents the name of the final output file. The **Delete Before Processing** checkbox is checked automatically to remove these files before processing begins. This prevents problems arising from the existence of an old file.



Batch Wizard : Edit Commands / Create Variables

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Name: Output

Type: Auto

Pattern: /usr/data/$(Input.root) image2.

☑ Delete Before Processing

☐ Delete After Processing

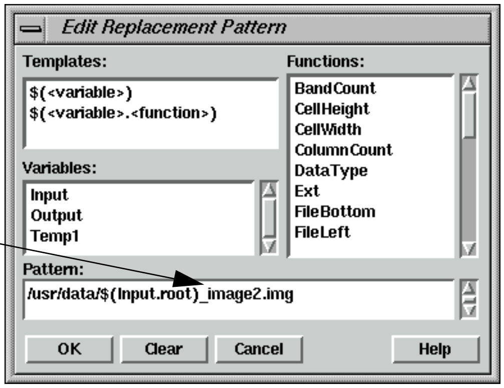
The Output file is also automatically generated

Use the Set button to edit the replacement pattern, and give the Output a different name

Notice that the **Output** variable as well as the **Temp1** variable described above are of the **Type Auto**. This means the software is responsible for creating the file names that are substituted for the variables at run time.

By default, the software uses the output directory that you specified in step 2. under "Execute Multiple Files/Multiple Commands" on page 196. For the file name, it uses a combination of the root name of the input file and the last specified output file (in this example, **image2**).

4. Click the **Set** button. Click in the **Pattern** window and edit the text string to change **image2** to **inverse**, and then click **OK**.


Edit Replacement Pattern
Templates:
$(<variable>)
$(<variable>.<function>)
Functions:
BandCount
CellHeight
CellWidth
ColumnCount
DataType
Ext
FileBottom
FileLeft
Variables:
Input
Output
Temp1
OK
Clear
Cancel
Help

Pattern: /usr/data/$(Input.root)_image2.img

Change this portion of the Pattern to the new name: inverse

NOTE: If your input files have the same root name (e.g., **03807701.nec**, **03807701.nwc**) *change* **Input.root** *to* **Input.name** *in the* **Pattern** *window. This includes the extension and thus preserves the uniqueness of the file name.*

Select Input Files

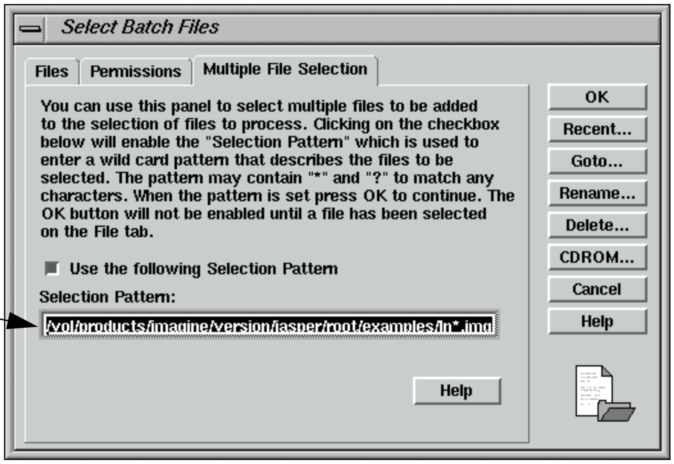
1. Click the **Next >** button to open the **Select Files to Process** panel. The original input file is listed in the **Input** column.

2. Click the Add Files button. The Select Batch Files dialog opens.

3. Select **lnaspect.img** from the Select Batch Files dialog. Click the **Multiple File Selection** tab and click the **Use the following Selection Pattern** checkbox. Edit the **Selection Pattern** field to insert **ln** before the asterisk. It should look like this:

```
<IMAGINE_HOME>/examples/ln*.img
```

Where <IMAGINE_HOME> represents the ERDAS IMAGINE installation directory.


Select Batch Files
Files
Permissions
Multiple File Selection
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
Help

You can use this panel to select multiple files to be added to the selection of files to process. Clicking on the checkbox below will enable the "Selection Pattern" which is used to enter a wild card pattern that describes the files to be selected. The pattern may contain "*" and "?" to match any characters. When the pattern is set press OK to continue. The OK button will not be enabled until a file has been selected on the File tab.

■ Use the following Selection Pattern

Selection Pattern: /vol/products/imagine/version/jasper/root/examples/ln*.img

Insert ln in the Selection Pattern window to select all files in the directory starting with the letters ln

4. Click the **OK** button on the Select Batch Files dialog.

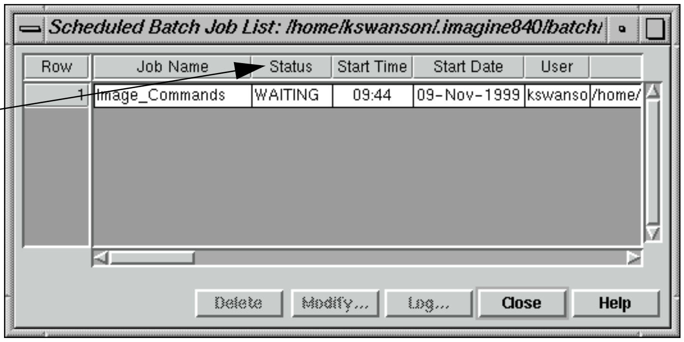
   The input files are displayed under the **Input** column. Select and delete any duplicate or unwanted files by clicking in the numbered column next to the file name and then clicking the Delete Files icon. This list of input files may be saved for future use.

   See “Additional Information” *on page 207 for details on how to save lists of input files.*

Set Start Time

1. Click the **Next >** button to open the **Select When to Process Commands** panel. Click the **Start Processing Later At** checkbox. Use the default time of the next available minute.

2. Click the **Finish** button to begin processing. The Scheduled Batch Job List dialog opens, displaying all current Batch jobs.


Scheduled Batch Job List: /home/kswanson/.imagine840/batch/
Delete
Modify...
Log...
Close
Help

Row	Job Name	Status	Start Time	Start Date	User	
1	Image_Commands	WAITING	09:44	09-Nov-1999	kswanso	/home/

The Status of the Batch job is still WAITING

When the job begins, the **Status** changes from **WAITING** to **ACTIVE**. When the job has finished, the **Status** changes from **ACTIVE** to **DONE**.

When the job is done, the log file can be viewed by first selecting the job and then clicking the **Log** button in the Scheduled Batch Job List dialog.

The Scheduled Batch Job List dialog may be opened any time by selecting **Session | View Off-line Batch Queue** from the ERDAS IMAGINE menu bar.

3. When the Batch jobs are finished, indicated by **DONE** in the **Status** column, click **Close** in the Scheduled Batch Job List dialog.

If you wish, you can check the files you just generated in a Viewer.

NOTE: You may also wish to change your preferences back to the default of **Compute Pyramid Layers** *in the* **Image Files (General)** *category of the Preference Editor.*

Work with Variables

This section deals with modification of automatically created variables for the purpose of generalizing a command file.

Create a New Variable

A new variable is helpful in the case of an application requiring multiple inputs. The automatically created variables always consist of a single **Input**, a single **Output**, and as many **Temp** variables as necessary. If you are creating a Batch command file that can be used by others, it is much more readable if the **Temp** variables are replaced by variable names that are meaningful.

When creating variables, it is important to know the distinction between **User** variables and **Auto** variables. The value of a **User** variable is derived from the **Select Files to Process** panel. You must provide the appropriate value for each **User** variable in the CellArray.

The value of an **Auto** variable is generated by the software from parameters specified in the pattern window of the **Set** dialog. If you recall, you worked with the dialog generated by pressing the **Set** button, the Edit Replacement Pattern dialog in "Modify Variables" on page 198.

In the following example, you are going to create a variable, **OutputDir**, which enables another person to enter an output directory of his or her choice for files processed by Batch. For this exercise, you have selected a number of files that you want to reproject.

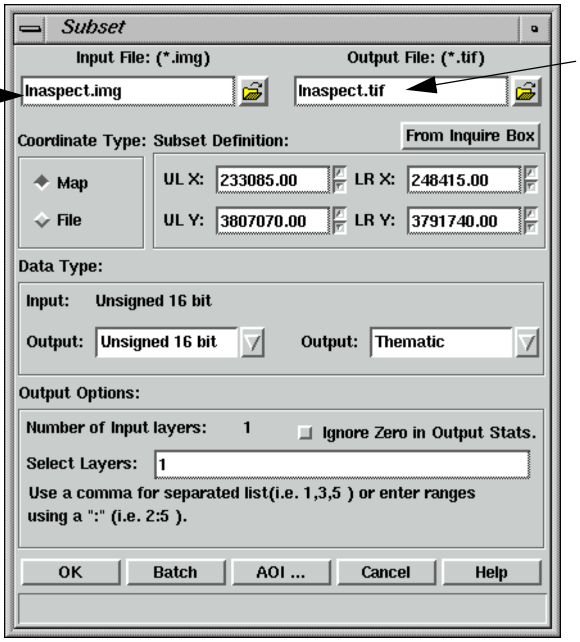
Prepare

First, you have to start the application from which you launch the Batch Wizard.

ERDAS IMAGINE must be running.

1. Click the Interpreter icon. From the **Image Interpreter** menu, select **Utilities**. From the **Utilities** menu, select **Subset**.

The Subset dialog opens.



Input File: (*.img) Inaspect.img

Output File: (*.tif) Inaspect.tif

Coordinate Type: ◆ Map ◇ File

Subset Definition:

UL X: 233085.00 LR X: 248415.00

UL Y: 3807070.00 LR Y: 3791740.00

Data Type:

Input: Unsigned 16 bit

Output: Unsigned 16 bit Output: Thematic

Output Options:

Number of Input layers: 1 ☐ Ignore Zero in Output Stats.

Select Layers: 1

Use a comma for separated list(i.e. 1,3,5) or enter ranges using a ":" (i.e. 2:5).

Select Inaspect.img as the Input File

You can use the Subset dialog to generate TIFF files

2. In the Subset dialog, click the Open icon underneath **Input File**. In the Input File dialog, navigate to the <IMAGINE_HOME>/examples directory. Choose the file **Inaspect.img** from the /examples directory. Click **OK** in the Input File dialog to transfer the file to the Subset dialog.

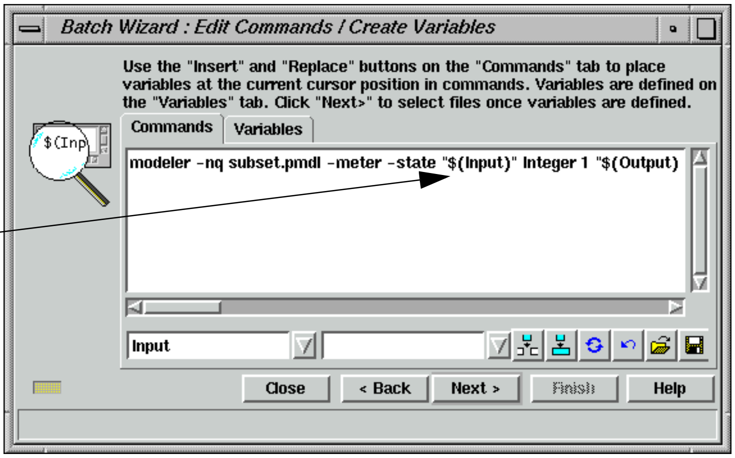
3. Click the Open icon underneath **Output File**. In the Output File dialog, navigate to the directory of your choice. Click the **File Type** dropdown list and select **TIFF**. In the **Filename** window, type **Inaspect**, then press Return on your keyboard. The .tif extension is appended automatically. Click **OK** in the Output File dialog.

Start Batch Wizard

1. Click the **Batch** button in the Subset dialog.

2. In the **Select Type of Command Processing** panel of the Batch Wizard dialog, click the **Modify commands automatically** checkbox. Then, click **Next >**.

   The **Edit Commands/Create Variables** panel displays.


Batch Wizard : Edit Commands / Create Variables
Commands
Variables
Close
< Back
Next >
Finish
Help

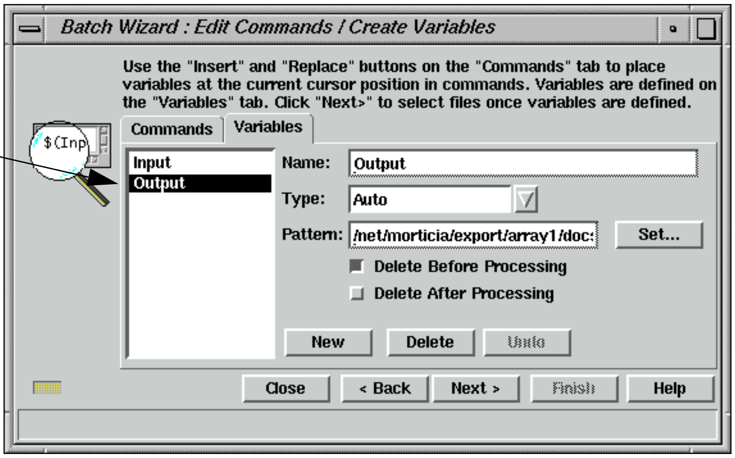
Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

```
modeler -nq subset.pmdl -meter -state "$(Input)" Integer 1 "$(Output)
```

Input

The Input and Output variables are automatically assigned

3. Click the **Variables** tab on the **Edit Commands/Create Variables** panel. Click on the **Output** variable to observe its properties.


Batch Wizard : Edit Commands / Create Variables
Commands
Variables
Input
Output
Set...
New
Delete
Undo
Close
< Back
Next >
Finish
Help

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Name: Output

Type: Auto

Pattern: /net/morticia/export/array1/doc:

☒ Delete Before Processing

☐ Delete After Processing

Click the Output variable to observe its properties

Notice in the **Pattern** window of the **Variables** tab that the path to the **Output** file is so long that it does not completely fit within it. This directory can be replaced with a new variable called **OutputDir**. You can set the **Pattern** of the **OutputDir** variable to any directory you choose. This makes your command shorter.

Create a New Variable

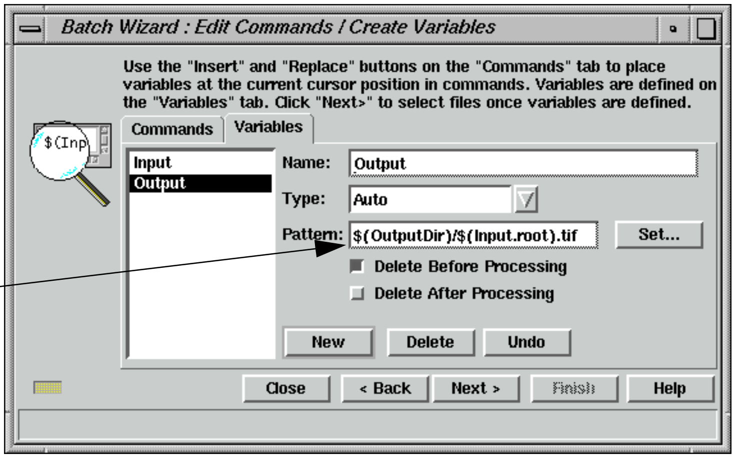
Since you know that you want your new variable to be named **OutputDir**, and you know you want it to replace the current path in the **Pattern** of the **Output** variable with it, you can begin editing there.

1. Make sure that the **Variables** tab displays the **Output** variable's details.

2. Click in the **Pattern** window and remove the entire directory structure you want to replace. Once you have eliminated it, type in the following:

```
$(OutputDir)
```

Your **Pattern** window then looks like the following:



Batch Wizard : Edit Commands / Create Variables

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Name: Output

Type: Auto

Pattern: $(OutputDir)/$(Input.root).tif

☒ Delete Before Processing

☐ Delete After Processing

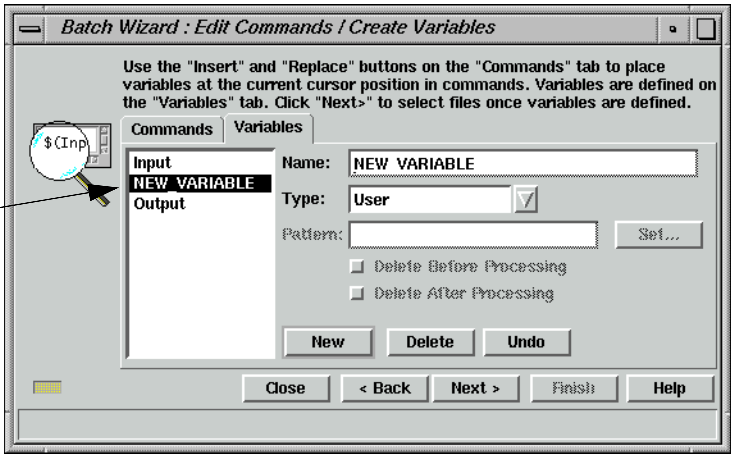
$(Inp
Commands
Variables
Input
Output
Set...
New
Delete
Undo
Close
< Back
Next >
Finish
Help

Replace the original path with the new variable OutputDir

You have to create the new variable, **OutputDir**, so that the variable **Output** does not generate errors in your Batch script, and knows where to place output files.

3. In the **Variables** tab of the **Edit Commands/Create Variables** panel, click the **New** button.

   The **Variables** tab changes to accommodate a new variable.



Batch Wizard : Edit Commands / Create Variables

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Name: NEW VARIABLE

Type: User

Pattern:

☐ Delete Before Processing

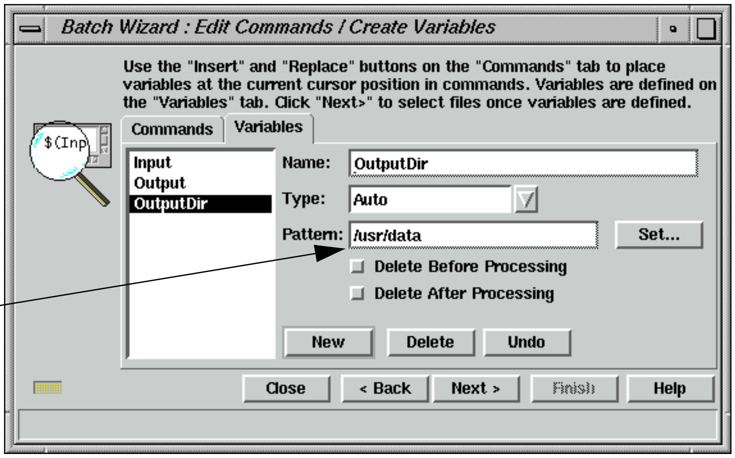
☐ Delete After Processing

$(Inp
Commands
Variables
Input
NEW VARIABLE
Output
Set...
New
Delete
Undo
Close
< Back
Next >
Finish
Help

The new variable is added to the list; now you supply properties

4. In the **Name** section of the **Variables** tab, type the name **OutputDir**.
5. Click the **Type** dropdown list and choose **Auto**.
6. In the **Pattern** window, type the path to the directory and folder that you want generated files to be saved in.

   Your **Variables** tab looks like the following:



Batch Wizard : Edit Commands / Create Variables

Use the "Insert" and "Replace" buttons on the "Commands" tab to place variables at the current cursor position in commands. Variables are defined on the "Variables" tab. Click "Next>" to select files once variables are defined.

Input
Output
OutputDir

Name: OutputDir
Type: Auto
Pattern: /usr/data

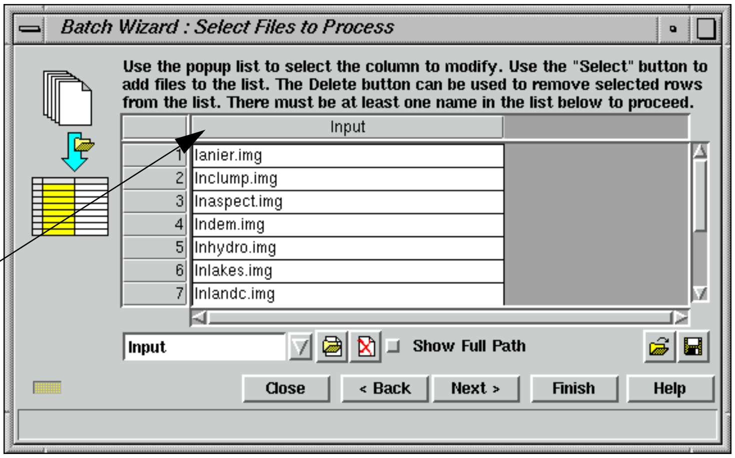
☐ Delete Before Processing
☐ Delete After Processing

All output files are going to be stored in the output directory /usr/data

Add Additional Files to the Batch Job

1. Click **Next >** in the **Edit Commands/Create Variables** panel.
2. In the **Select Files to Process** panel, click the Select Files to Add button, and add all files with in the <IMAGINE_HOME>/examples directory that start with the letters **ln** using the **Multiple File Selection** tab.

If you need to review how to add multiple files, see "Add Multiple Files" on page 190.



Batch Wizard : Select Files to Process

Use the popup list to select the column to modify. Use the "Select" button to add files to the list. The Delete button can be used to remove selected rows from the list. There must be at least one name in the list below to proceed.

	Input
1	lanier.img
2	lnclump.img
3	lnaspect.img
4	lndem.img
5	lnhydro.img
6	lnlakes.img
7	lnlandc.img

Input

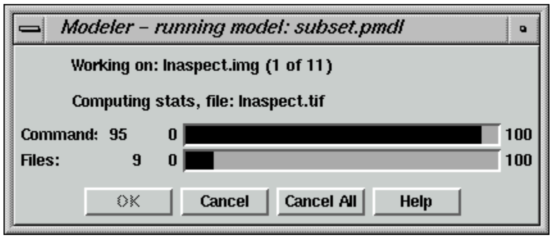
☐ Show Full Path

These are all the files that begin with ln and are located in the directory identified in the Selection Pattern window

Start Processing

1. Click the **Next >** button to open the **Select When to Process Commands** panel.
2. Make sure the **Start Processing Now** option is selected, then click **Finish** to start the Batch job with the new variable.

The double Job Status dialog opens.



Working on: lnaspect.img (1 of 11)

Computing stats, file: lnaspect.tif

Command: 95

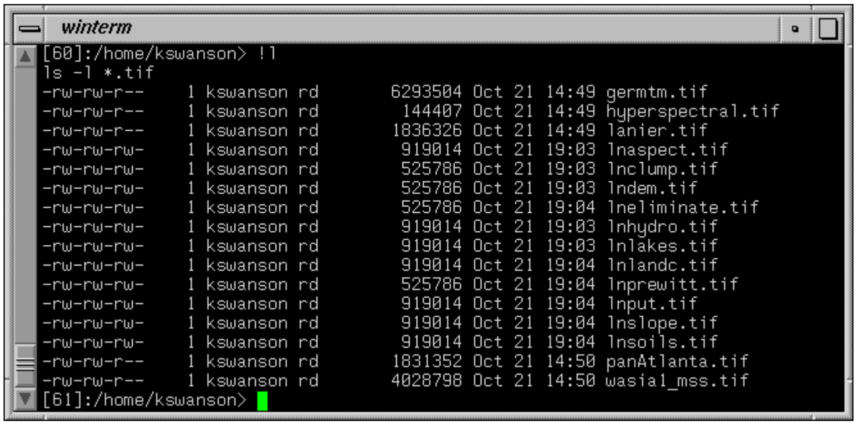
Files: 9

3. When the process completes, click **OK** in the Job Status dialog.

Check the Output Directory

Now, you can check the output directory to see the files that have been stored there. In this example, the directory holds TIFF files you created from image files using the Subset utility of Image Interpreter.

1. Using a UNIX shell or Microsoft Explorer, navigate to the directory you specified as the Output Directory.
2. Check for the new TIFF files created from all files starting with **ln** in the <IMAGINE_HOME>/examples directory.



```
[60]:/home/kswanson> !l
ls -l *.tif
-rw-rw-r--    1 kswanson rd       6293504 Oct 21 14:49 germtm.tif
-rw-rw-r--    1 kswanson rd        144407 Oct 21 14:49 hyperspectral.tif
-rw-rw-r--    1 kswanson rd       1836326 Oct 21 14:49 lanier.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:03 lnaspect.tif
-rw-rw-rw-    1 kswanson rd        525786 Oct 21 19:03 lnclump.tif
-rw-rw-rw-    1 kswanson rd        525786 Oct 21 19:03 lndem.tif
-rw-rw-rw-    1 kswanson rd        525786 Oct 21 19:04 lneliminate.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:03 lnhydro.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:03 lnlakes.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:04 lnlandc.tif
-rw-rw-rw-    1 kswanson rd        525786 Oct 21 19:04 lnprewitt.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:04 lnput.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:04 lnslope.tif
-rw-rw-rw-    1 kswanson rd        919014 Oct 21 19:04 lnsoils.tif
-rw-rw-r--    1 kswanson rd       1831352 Oct 21 14:50 panAtlanta.tif
-rw-rw-r--    1 kswanson rd       4028798 Oct 21 14:50 wasia1_mss.tif
[61]:/home/kswanson> 
```

Additional Information

Save/Load Options

The **Edit Commands / Create Variables** panel and the **Select Files to Process** panel provide methods for saving and loading the Batch commands and input file lists, respectively.

Save

Click the Save icon to save the commands in the **Edit Commands/Create Variables** panel to a text (.bcf) file, and the input file list in the **Select Files to Process** panel to a text (.bls) file for later use. The Save Batch Commands dialog opens, allowing you to enter a new file name.

Load

Click the Load icon to load a previously saved command (.bcf) file or input file list (.bls) file. A Load Batch Commands dialog opens, allowing you to select from a previously saved list of files. These lists are text files that may be created outside of ERDAS IMAGINE.

Batch Job Files

Several files are created for each Batch job that runs at a later time. The default file name root is **batch_job**. The file name can be seen in the **Name** field of the **Select When to Process Commands** panel. File names for each Batch job are listed in the Scheduled Batch Job List dialog.

PC Version

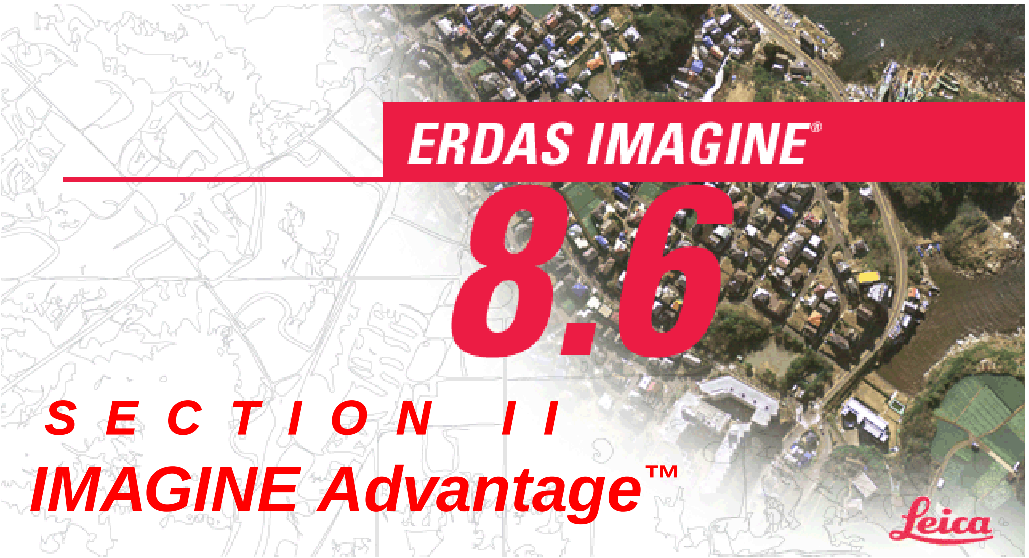
The following are PC versions of Batch job files that are located in the Batch job directory. The default Batch job directory is **$PERSONAL/batch**, and you can go to **Session | Preferences | Batch Processing** to change the **Batch Job Directory**.

- **batch_job.bat**—This is an MS-DOS Batch file. This file sets up the environment for the ERDAS IMAGINE applications and runs “batchprocess,” which runs the commands found in the **batch_job.bcf** file.
- **batch_job.bcf**—These are ERDAS IMAGINE Batch command files. These files hold the variable definitions and issue the ERDAS IMAGINE commands to perform the step, or steps, required to complete a job.
- **batch_job.bls**—These files contain lists of input files.
- **batch_job.id**—This file contains a job number assigned by the system. This file is deleted automatically to indicate completion of the job.
- **batch_job.lck**—This is an empty file that is created when the job actually starts running. This indicates the job is **ACTIVE**. When the job is done, it is deleted.
- **batch_job.log**—This file contains the results of the Batch job.

UNIX Version

These are UNIX versions of Batch job files:

- **batch_job.bcf**—These are ERDAS IMAGINE Batch command files. These files hold the variable definitions and issue the ERDAS IMAGINE commands to perform the step, or steps, required to complete a job.
- **batch_job.bls**—These files contain lists of input files.
- **batch_job.id**—This file contains a job number assigned by the system. This file is deleted automatically to indicate completion of the job.
- **batch_job.job**—This is a UNIX shell script. This file sets up the environment for the ERDAS IMAGINE applications and runs "batchprocess," which runs the commands found in the **batch_job.bcf** file.
- **batch_job.lck**—This is an empty file that is created when the job actually starts running. This indicates the job is **ACTIVE**. When the job is done, it is deleted.
- **batch_job.log**—This file contains the results of the Batch job.


ERDAS IMAGINE®
8.6
Leica

SECTION II

IMAGINE Advantage™

CHAPTER TEN

Fourier Transform Editor

Introduction

In this tour guide, you enhance and destripe a 512 × 512 subset of a Landsat Thematic Mapper image using both interactive and automatic methods available in the ERDAS IMAGINE Fourier Analysis tools.

Not all of the edits in this tour guide necessarily enhance the image. Many exercises are performed simply to show you how they affect the image. When you use these techniques on other data sets, you may want to experiment with different methods, or combinations of methods, to find the techniques that work best.

It is highly recommended that you read the Fourier Analysis section, beginning on page 171 of the ERDAS Field Guide, *before going through this tour guide.*

If you are a new to ERDAS IMAGINE, it is recommended that you go through the tour guide "Chapter 1: Viewer" *before using the Fourier Transform Editor.*

Approximate completion time for this tour guide is 45 minutes.

Create an .fft Layer

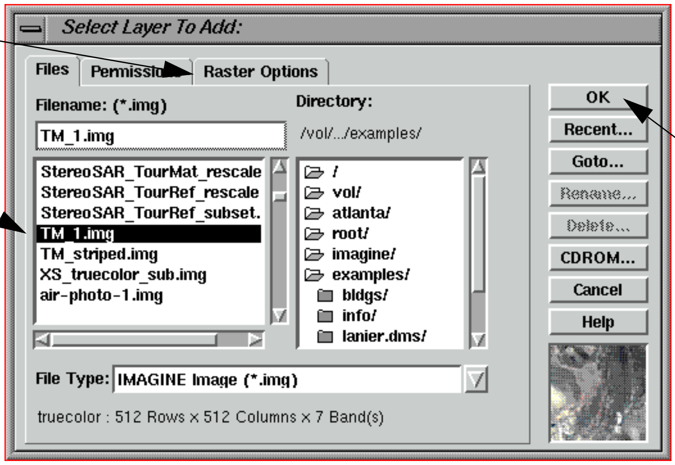
In order to use the Fourier Transform Editor, you must first create a Fourier Transform (.fft) layer from the input image.

Display Source File

ERDAS IMAGINE must be running and you must have a Viewer open.

1. Select **File | Open | Raster Layer** from the Viewer menu bar.

   The Select Layer To Add dialog opens.



Click here to view the raster options

Click here to select the file TM_1.img

Click here to display the file

Filename: (*.img) TM_1.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

truecolor : 512 Rows × 512 Columns × 7 Band(s)

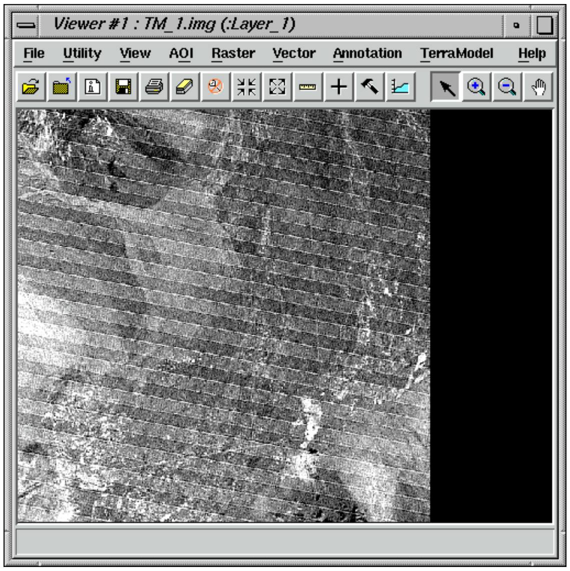
Select Layer To Add:
Files
Permission
Raster Options
StereoSAR_TourMat_rescale
StereoSAR_TourRef_rescale
StereoSAR_TourRef_subset.
TM_1.img
TM_striped.img
XS_truecolor_sub.img
air-photo-1.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

2. In the Select Layer To Add dialog under **Filename**, click on **TM_1.img**.

3. Click on the **Raster Options** tab at the top of the dialog and then select **Gray Scale** from the **Display as** popup list.

4. The **Display Layer** section updates so that you can select which layer of the file to display. Accept the default of **Layer 1**.

5. Click **OK** to display the image file in the Viewer.

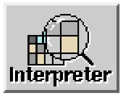


Viewer #1 : TM_1.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

NOTE: You do not have to display a file before computing the .fft layer. This step is included to show you the image before any Fourier editing is performed.

Create FFT Output File

1. Click the Image Interpreter icon on the ERDAS IMAGINE icon panel.


Interpreter

The **Image Interpreter** menu displays.

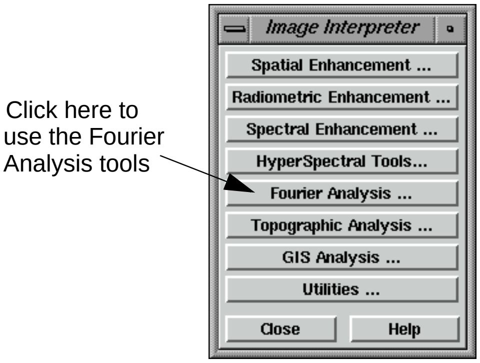
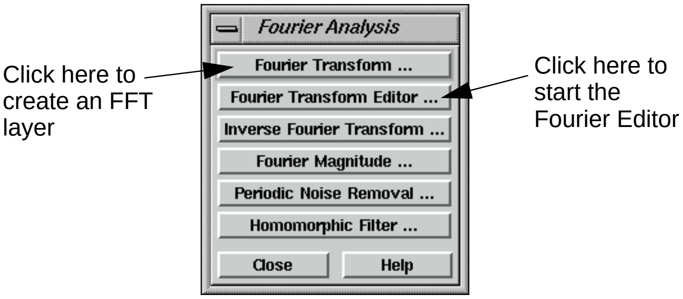


Image Interpreter
Spatial Enhancement ...
Radiometric Enhancement ...
Spectral Enhancement ...
HyperSpectral Tools...
Fourier Analysis ...
Topographic Analysis ...
GIS Analysis ...
Utilities ...
Close
Help

Click here to use the Fourier Analysis tools

2. Select **Fourier Analysis** from the **Image Interpreter** menu.

The **Fourier Analysis** menu opens.

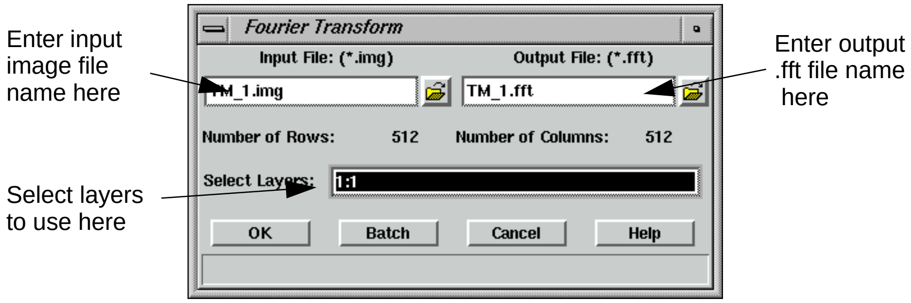

Fourier Analysis
Fourier Transform ...
Fourier Transform Editor ...
Inverse Fourier Transform ...
Fourier Magnitude ...
Periodic Noise Removal ...
Homomorphic Filter ...
Close
Help

Click here to create an FFT layer

Click here to start the Fourier Editor

3. Select **Fourier Transform** from the **Fourier Analysis** menu.

The Fourier Transform dialog opens.



Enter input image file name here

Select layers to use here

Enter output .fft file name here

Input File: (*.img) TM_1.img

Output File: (*.fft) TM_1.fft

Number of Rows: 512

Number of Columns: 512

Select Layers: 1:1

4. In the Fourier Transform dialog under **Input File**, type in **TM_1.img**.

   Layer 1 of this file is badly striped. In this example, you work with only one layer to make the processing go faster. However, the techniques you use are applicable to multiple layers.

5. The name for the **Output File**, **TM_1.fft**, is automatically generated. Make sure it is in a directory in which you have write permission. The default is your default data directory set by **Session | Preferences**.

6. Enter **1:1** in the **Select Layers** field.

7. Click **OK** in the Fourier Transform dialog to create the new file.

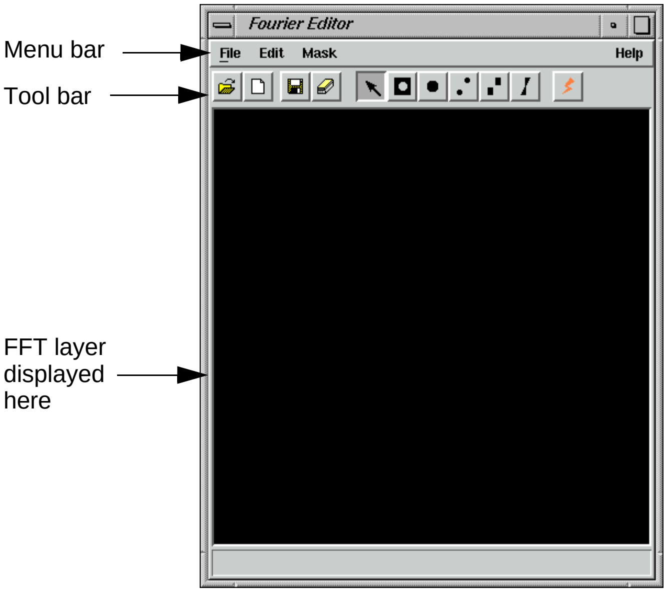
   A Job Status dialog displays, showing the progress of the function. When the process is 100 percent complete, click **OK**.

Start the Fourier Transform Editor

With the .fft file created, you are ready to begin using the Fourier Transform Editor.

1. In the **Fourier Analysis** menu, select **Fourier Transform Editor**.

   The Fourier Editor opens.


Fourier Editor
File
Edit
Mask
Help

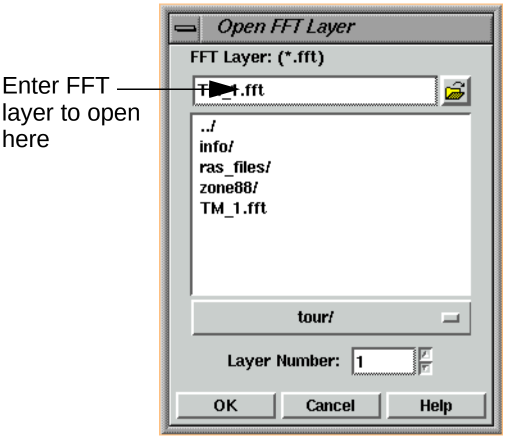
Menu bar

Tool bar

FFT layer displayed here

2. In the Fourier Editor, click the Open icon on the tool bar, or select **File | Open** from the menu bar.

The Open FFT Layer dialog opens.


Open FFT Layer
../
info/
ras_files/
zone88/
TM_1.fft
tour/
OK
Cancel
Help

FFT Layer: (*.fft) TM_1.fft

Layer Number: 1

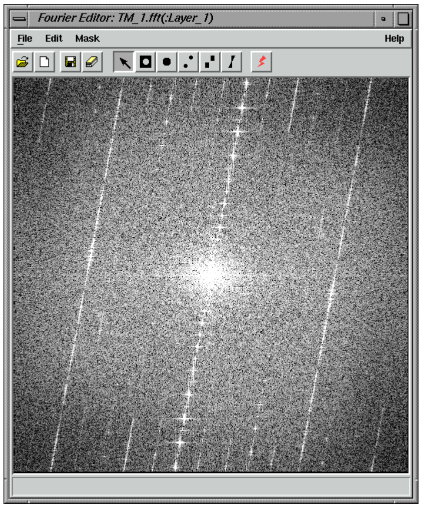
Enter FFT layer to open here

3. In the Open FFT Layer dialog under **FFT Layer**, enter the path and name of the .fft layer you created in steps (4. through (5. on page 216 (e.g., **TM_1.fft**).

Since this file contains only one layer, the **Layer Number** defaults to **1**. However, if the file contained more than one layer, you could choose the layer to edit here. Edits performed on one layer can be applied to all layers of the .fft file using the **File | Save All** option on the Fourier Editor menu bar.

4. Click **OK** to display the selected file in the Fourier Editor.

   A status meter opens as the layer is read. Then the layer is displayed.

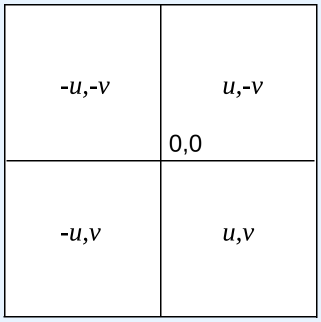


You can resize the Fourier Editor window to see the entire file.

5. Click on any point inside the Fourier Editor and the coordinates of that point are shown in the status bar. Hold and drag to dynamically update the coordinates.

Fourier Editor Coordinates

The coordinates are referred to as (*u*,*v)* with the origin (*u*,*v* = 0,0) at the center of the image. See the illustration below.



Since Fourier images are symmetrical, a point in one quadrant is exactly the same as the corresponding point in the opposite quadrant. For example, point (64,170) is the same as point (-64,-170). For this reason, all edits are automatically performed on both halves of the image at the same time.

Edit Using Menu Options

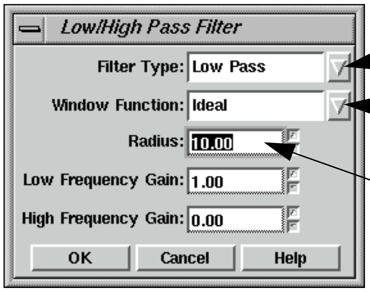
As previously stated, the menu bar and mouse-driven tools offer the same techniques and kinds of edits, only the method is different. In the menu bar options, you enter all parameters into dialogs. In many cases you want to use the mouse to view the coordinates of the .fft layer, so that you know what information to enter into the dialogs. In the next series of steps, you use some of the menu bar editing options. Then, in the next section, you perform many of those same edits using the mouse-driven tools.

Use Low-Pass Filtering

Low-pass filtering allows you to attenuate the high-frequency components of the image, but allows the low-frequency components to pass through.

1. Select **Mask | Filters** from the Fourier Editor menu bar.

   The Low/High Pass Filter dialog opens.



Filter Type: Low Pass

Window Function: Ideal

Radius: 10.00

Low Frequency Gain: 1.00

High Frequency Gain: 0.00

Select filter to use here

Select window function here

Set window radius here

Filter Types

When the **Filter Type** is set to **Low Pass**, its function is the same as the Low Pass Filter icon on the tool bar.

When the **Filter Type** is set to **High Pass**, its function is the same as the High Pass Filter icon on the tool bar.

You use these mouse-driven tools later in this tour guide.

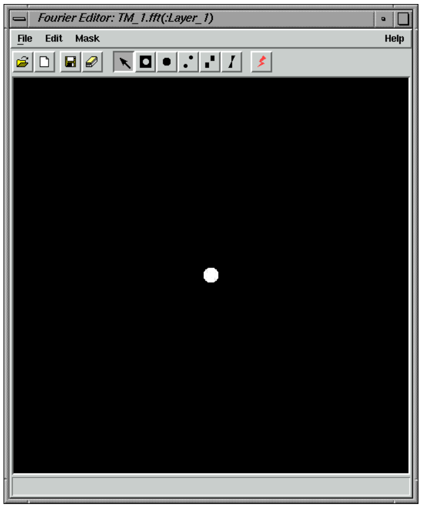
2. In the Low/High Pass Filter dialog, click on the **Window Function** popup list and select **Ideal**.

   An ideal window function produces a sharp transition at the edge of the filter.

3. Change the **Radius** to **10.00**.

4. Leave all other parameters as is and click **OK**.

   A low-pass filter is applied to all values outside of the radius of 10.00. Therefore, the image is black, except for a small white circle in the center.



Removing this much of the layer removes much of the content of the image, so you may want to undo this edit and try again.

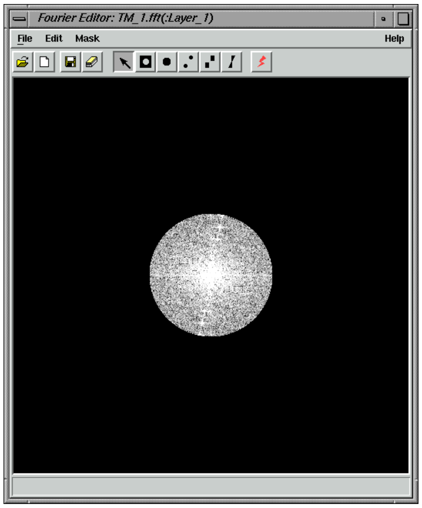
5. Select **Edit | Undo** from the Fourier Editor menu bar.

   The image is restored to its original state.

Select a Different Filter

1. Select **Mask | Filters**.
2. In the Low/High Pass Filter dialog, click on the **Window Function** popup list and select **Ideal**.
3. Enter a **Radius** of **80.00**.
4. Click **OK** in the Low/High Pass Filter dialog.

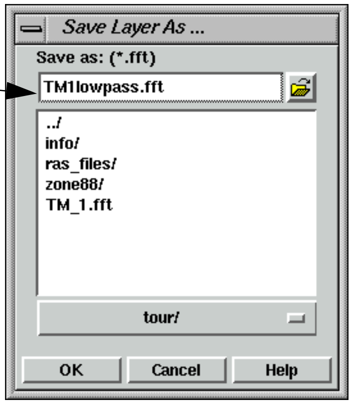
   All frequencies outside the radius of 80 are attenuated and frequencies inside the radius are unaffected. The .fft layer looks similar to the following example:



Save the File

1. Select **File | Save As** from the Fourier Editor menu bar.

   The Save Layer As dialog opens.



Save Layer As ...

Save as: (*.fft)

TM1lowpass.fft

../
info/
ras_files/
zone88/
TM_1.fft

tour/

Enter a new name for the FFT layer here

2. In the directory of your choice, enter a name for the new .fft layer, such as **TM1lowpass.fft**.

3. Click **OK** to save the file.

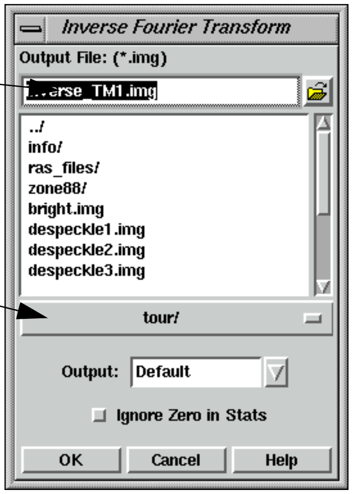
Apply an Inverse Fourier Transformation

Now, perform an inverse Fourier transformation so that you can view the original image and see what effect this edit had on it.

You must save your edits before performing an Inverse Transform Operation.

1. In the Fourier Editor, click the Run icon on the tool bar or select **File | Inverse Transform** from the menu bar.

The Inverse Fourier Transform dialog opens.



Inverse Fourier Transform

Output File: (*.img)

...erse_TM1.img

../
info/
ras_files/
zone88/
bright.img
despeckle1.img
despeckle2.img
despeckle3.img

tour/

Output: Default

☐ Ignore Zero in Stats

Enter a new name for the .img layer here

Be sure to use a directory in which you have write permission

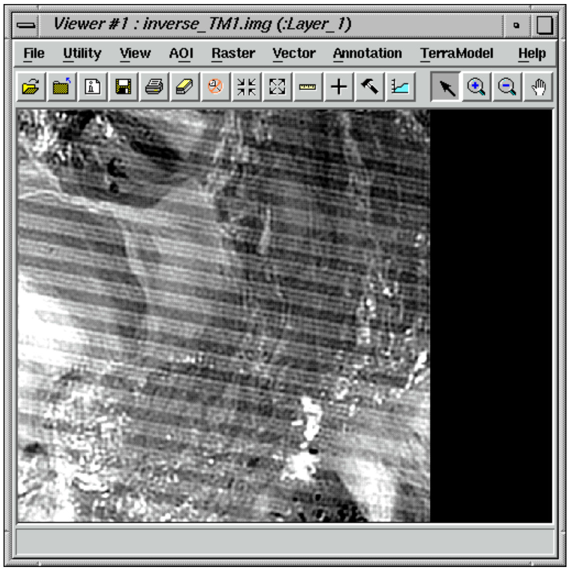
2. In the Inverse Fourier Transform dialog under **Output File**, enter a name for the new output file, such as **inverse_TM1.img**. This file has an .img extension by default. Be sure to use a directory in which you have write permission.

3. Click **OK** to create the new file.

   A Job Status dialog displays, indicating the progress of the function.

4. When the Job Status dialog indicates that the file is created, click **OK** and then display the file in a Viewer.

   Your file should look similar to the following example:



For the other edits performed in this tour guide, you can save the .fft layer and perform an inverse Fourier transform at any time. The steps for doing so are not repeated here. However, the result is shown so that you can see how each edit affects the image.

Apply Other Filters

1. In the Fourier Editor, click the Open icon on the tool bar, or select **File | Open** from the menu bar.

2. In the Open FFT Layer dialog under **FFT Layer**, enter the name of the first .fft layer you displayed (e.g., **TM_1.fft**).

3. Click **OK** to display the selected file in the Fourier Editor.

4. When the file displays, select **Mask | Filters** from the Fourier Editor menu bar.

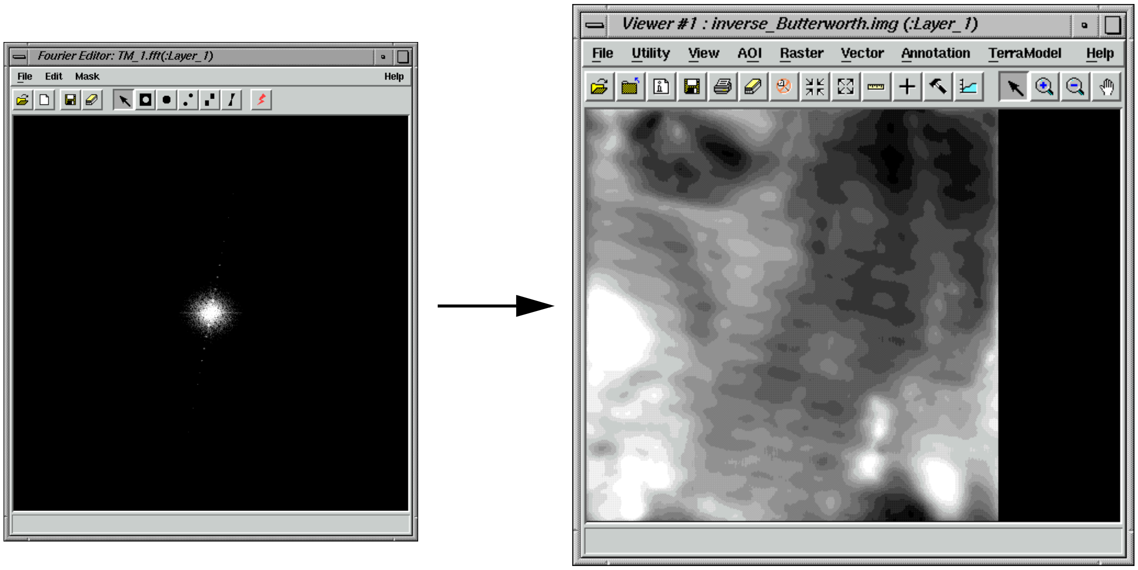
5. In the Low/High Pass Filter dialog, click on the **Window Function** popup list and select **Butterworth**.

   This is a smoother function than the Ideal. Use a radius of 80.00, just as with the Ideal.

6. Change the **Radius** to **80.00**.

7. Click **OK** in the Low/High Pass Filter dialog.

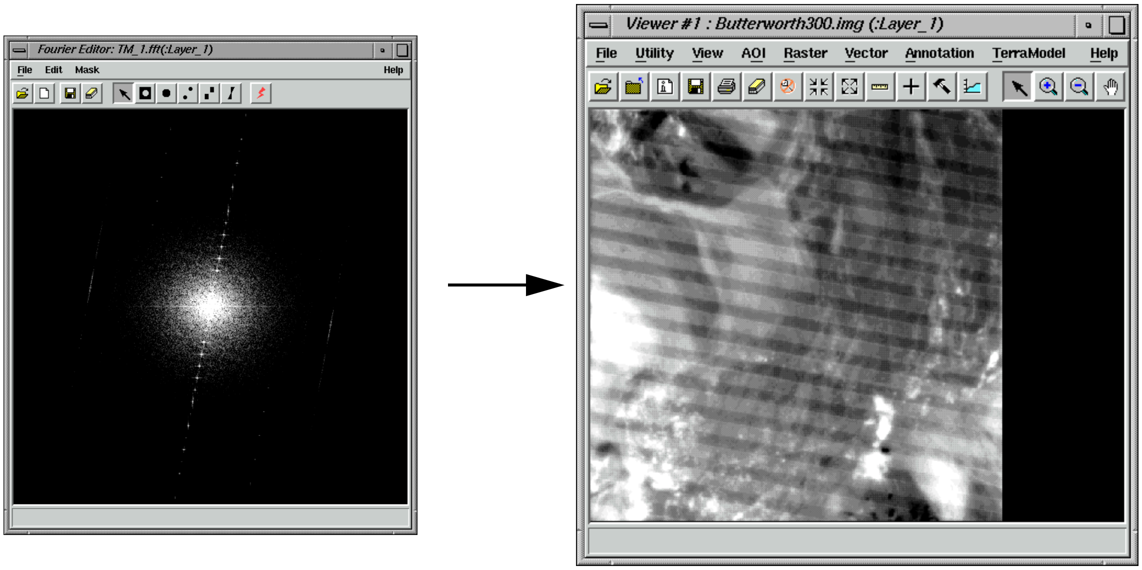
   The .fft layer and the resulting image are shown in the following picture:


Fourier Editor: TM_1.fft(:Layer_1)
File
Edit
Mask
Help
Viewer #1 : inverse_Butterworth.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

   This filter eliminated much of the image content because the radius was too small.

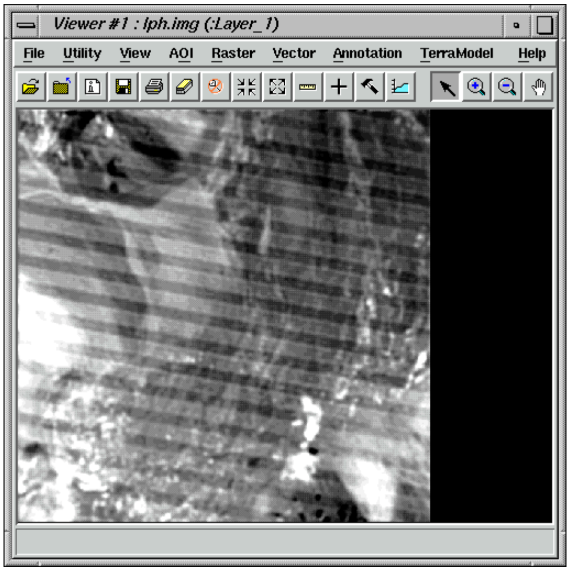
8. Try this same exercise using a **Radius** of **300.00**, rather than **80.00**

   The resulting image looks like the following example:

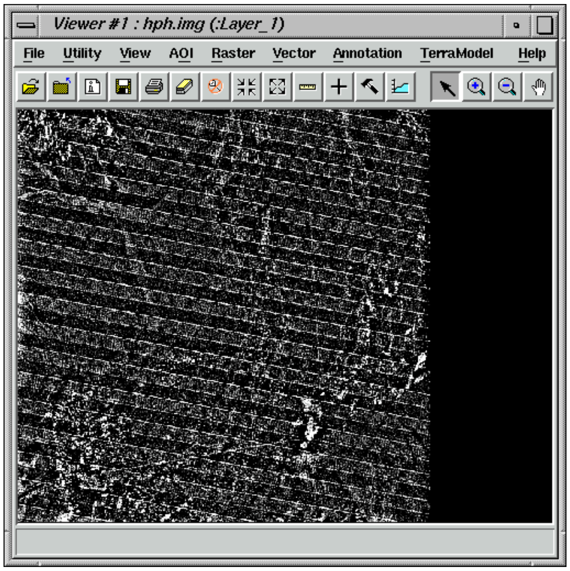

Fourier Editor: TM_1.fft(:Layer_1)
File Edit Mask Help
Viewer #1 : Butterworth300.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

The image is visibly smoothed (perhaps too much). However, the striping remains. You remove the stripes using the wedge filter later in this tour guide. You could try using the Butterworth filter with an even larger radius or the other windows.

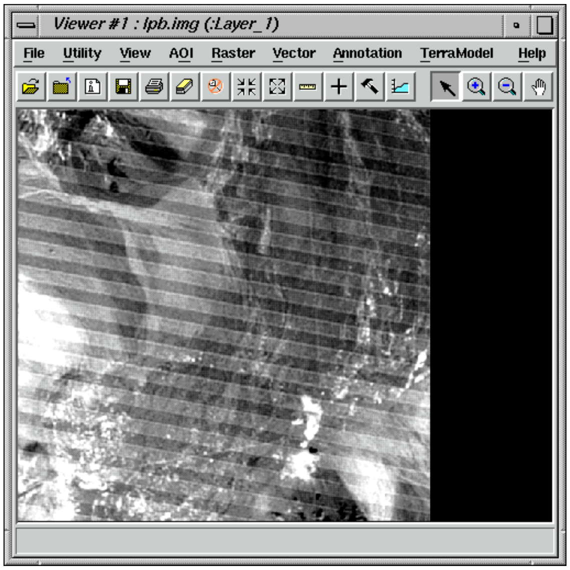
The following graphics illustrate some of these other scenarios.


Viewer #1 : lph.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

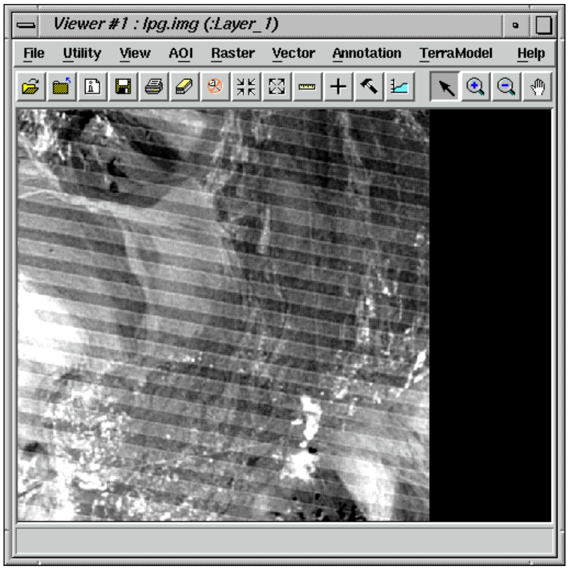
Type: Low Pass
Window: Hanning
Radius: 100.00


Viewer #1 : hph.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

Type: High Pass
Window: Hanning
Radius: 20.00


Viewer #1 : lpb.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Type: Low Pass
Window: Bartlett
Radius: 150.00


Viewer #1 : lpg.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Type: Low Pass
Window: Gaussian
Radius: 200.00

9. Redisplay the original .fft layer in the Fourier Editor.

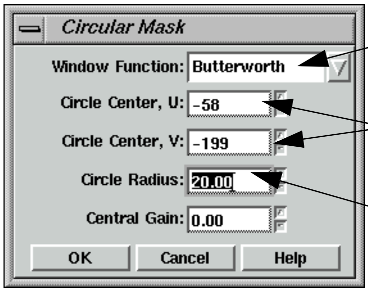
Use a Circular Mask

There are several bright spots in the .fft layer, such as those in the upper left quadrant. These can be eliminated using the circular mask option.

1. With your cursor in the Fourier Editor, click in the center of one of these bright areas. There is one at (*u*,*v*) = (-58,-199). You use this coordinate here, but you can use another if you like.

2. When you have selected a coordinate, select **Mask | Circular Mask** from the Fourier Editor menu bar.

   The Circular Mask dialog opens.


Circular Mask
OK
Cancel
Help

Window Function: Butterworth
Circle Center, U: -58
Circle Center, V: -199
Circle Radius: 20.00
Central Gain: 0.00

Select window function here

Enter center coordinate here

Enter radius here

This option is the same as if you were to click on the Circular Mask icon on the tool bar.

3. In the Circular Mask dialog, click on the **Window Function** popup list and select **Butterworth**.

4. Enter **-58** for the **Circle Center, U** and **-199** for the **Circle Center, V**.

5. Enter a **Circle Radius** of **20**.

6. Click **OK** in the Circular Mask dialog to edit the .fft layer.

   The bright spot disappears. This edit does not affect the appearance of this particular image very much, since it is such a small area and because the edited area is quite far from the center of the image where most of the image content is contained. However, this technique can be used to remove spikes caused by errant detectors and other types of periodic noise that are manifested by concentrated areas of high or low frequency in the .fft layer.

   As an experiment, you create two circles of low frequency to see how they affect the image.

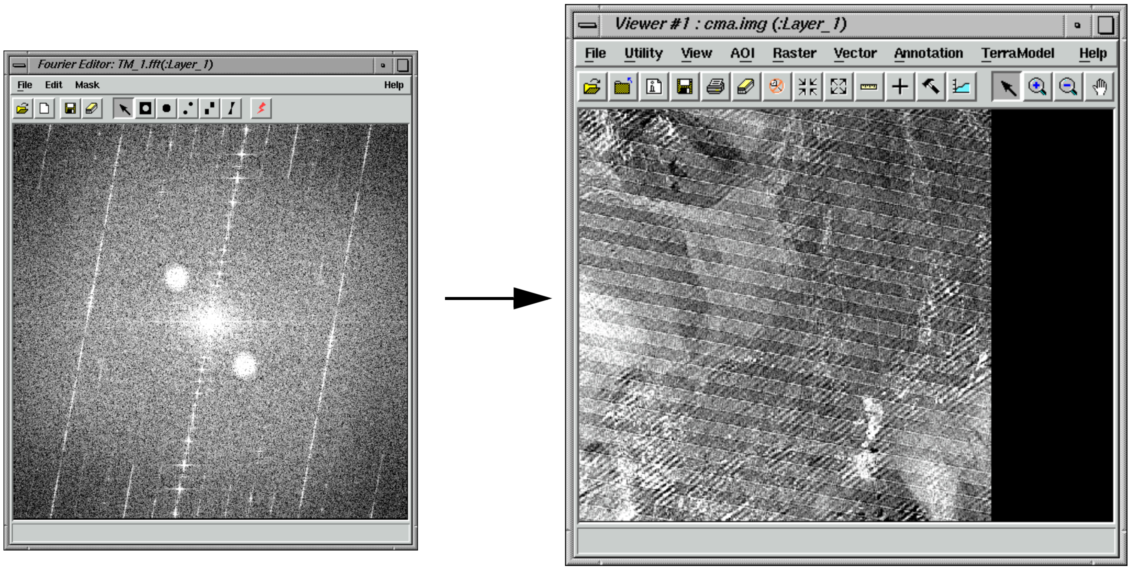
7. In the Fourier Editor, select **Mask | Circular Mask**.

8. In the Circular Mask dialog, enter a **Circle Center, U** of **44** and a **Circle Center, V** of **57**.

9. Enter a **Circle Radius** of **20.00** and a **Central Gain** of **10.00**.

10. Click **OK** in the Circular Mask dialog.

    The .fft layer and resulting image look like the following example:


Fourier Editor: TM_1.fft(:Layer_1)
File
Edit
Mask
Help
Viewer #1 : cma.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

The resulting image has a pronounced diagonal striping, in addition to the original striping.

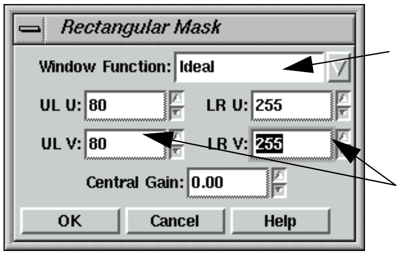
NOTE: Remember to select another circle center coordinate before trying each new window.

Use a Rectangular Mask

The rectangular mask allows you to mask a rectangular area of the .fft layer. This is similar to the circular mask in that it allows you to edit non-central regions of the Fourier image.

1. Make sure the .fft file you created in the previous section,"Use a Circular Mask", is displayed in the Fourier Editor.
2. In the Fourier Editor menu bar, select **Mask | Rectangular Mask**.

   The Rectangular Mask dialog opens.



Window Function: Ideal

UL U: 80

LR U: 255

UL V: 80

LR V: 255

Central Gain: 0.00

Select window function here

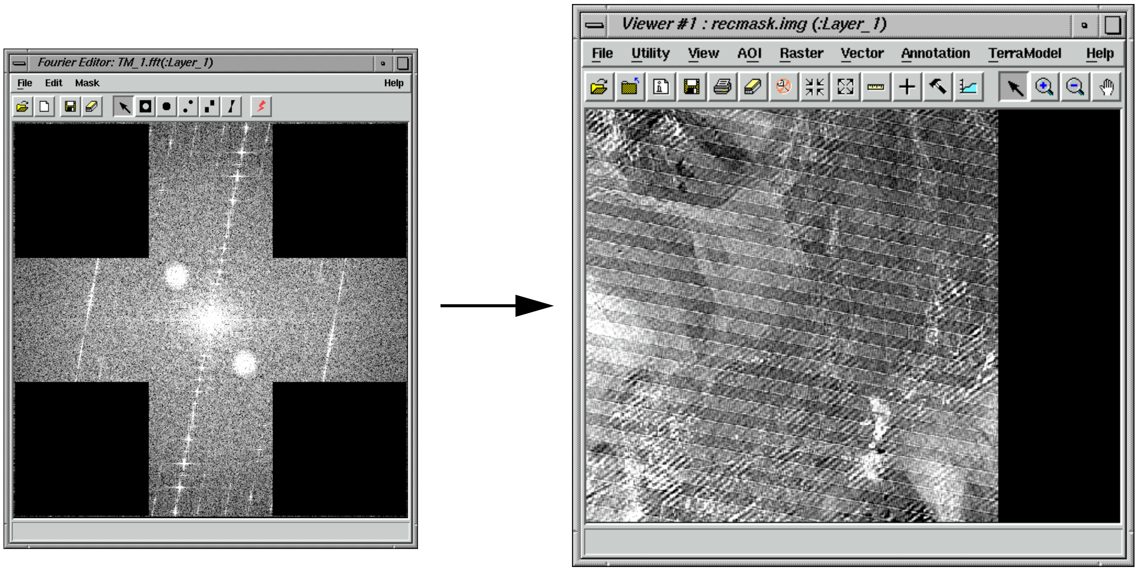
Enter coordinates of rectangle here

3. In the Rectangular Mask dialog, click on the **Window Function** popup list and select **Ideal**.
4. Enter an upper left *u* (**UL U**) of **80** and an upper left *v* (**UL V**) of **80**.
5. Enter a lower right *u* (**LR U**) of **255** and a lower right *v* (**LR V**) of **255**.
6. Click **OK** in the Rectangular Mask dialog.

   The top left and bottom right corners of the .fft layer are black. To mask the other two corners, you must repeat this procedure.

7. Select **Mask | Rectangular Mask**.
8. In the Rectangular Mask dialog, click on the **Window Function** popup list and select **Ideal**.
9. Enter an upper left *u* (**UL U**) of **80** and an upper left *v* (**UL V**) of **-255**.
10. Enter a lower right *u* (**LR U**) of **255** and a lower right *v* (**LR V**) of **-80**.
11. Click **OK** in the Rectangular Mask dialog.

    The top, right and bottom, left corners of the .fft layer are now black also, making the .fft layer look like a cross.


Fourier Editor: TM_1.fft(:Layer_1)
File Edit Mask Help
Viewer #1 : recmask.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

The resulting image is visibly smoother than the original.

Use a Wedge Mask

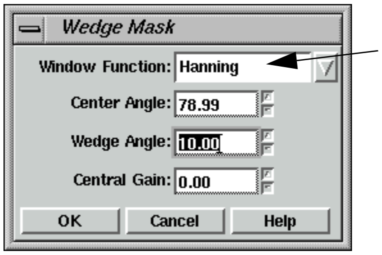
The wedge mask option is often used to remove striping in imagery that appears in the .fft layer as radial lines. Most of the striping in the Landsat image you are using is manifested in the .fft layer as the bright, nearly vertical line that passes through the origin.

1. If it is not already displayed, open the original .fft layer in the Fourier Editor (i.e., **TM_1.fft**).

2. With your cursor in the Fourier Editor, click in the center of one of the bright areas that make up the line. You need to enter this information in the dialog. For this example, you use (35, -186).

3. Select **Mask | Wedge Mask**.

   The Wedge Mask dialog opens.


Wedge Mask
OK
Cancel
Help

Window Function: Hanning

Center Angle: 78.99

Wedge Angle: 10.00

Central Gain: 0.00

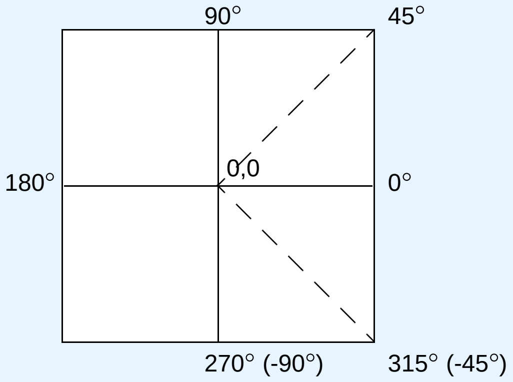
Select window function here

4. In the Wedge Mask dialog for the **Center Angle**, enter the following expression to calculate the center of the wedge, based on the coordinate that you selected.
   -atan(-185/36)

5. Press Return on your keyboard. The value returned is **78.99**.

Wedge Mask Angles

The angles are measured as shown in the illustration below.

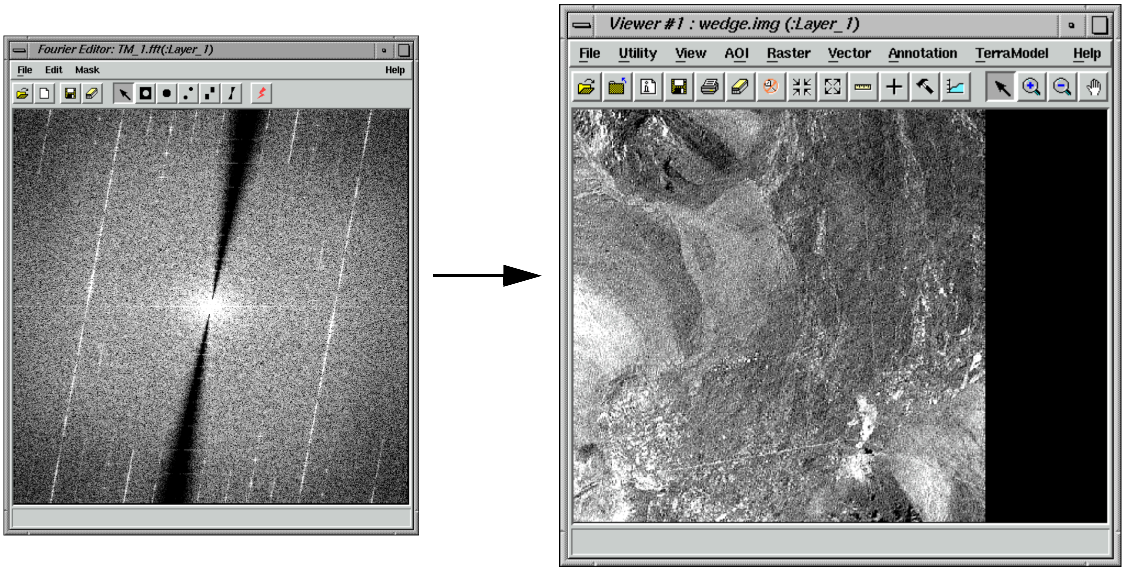

90°
45°
180°
0,0
0°
270° (-90°)
315° (-45°)

6. Enter a **Wedge Angle** of **10.00**.

   This is the total angle of the wedge, in this case, 5.00 degrees on either side of the center.

7. Click **OK** to edit the layer.

   The resulting .fft layer looks similar to the following example:


Fourier Editor: TM_1.fft(:Layer_1)
File Edit Mask
Help
Viewer #1 : wedge.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

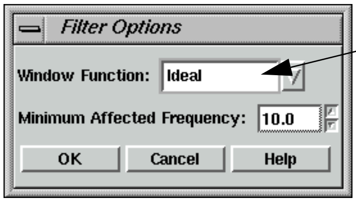
After performing an inverse Fourier transform, the resulting image is destriped.

Edit Using Mouse-Driven Tools

The mouse-driven tools allow you to perform the same types of edits as in the menu options, but they are a bit easier to use since they are more interactive than the dialogs. You can extend a filter radius or indicate where to place a mask simply by dragging the mouse.

1. If it is not already displayed, open the original .fft layer in the Fourier Editor (e.g., **TM_1.fft**).
2. From the Fourier Editor menu bar, select **Edit | Filter Options**.

   The Filter Options dialog opens.



Window Function: Ideal

Minimum Affected Frequency: 10.0

Select window function here

This is where you set the window that is used for all subsequent mouse-driven editing options. However, you can change this window at any time. The **Minimum Affected Frequency** option allows you to enter the minimum frequency value that is affected by the filter. Setting this value to a number less than 10.00 might eliminate very low frequency data that are crucial to the content of the image.

3. In the Filter Options dialog, click on the **Window Function** popup list and select **Ideal**.
4. Click **OK**.

Use Low-Pass Filtering

The first tool you use is the Low-Pass Filter tool.

1. Click the Low-Pass Filter icon on the Fourier Editor tool bar.
2. With your cursor in the center of the Fourier Editor, drag toward the right until the *u* coordinate in the status bar reads **80**. Then release the mouse.

   The image is filtered as soon as the mouse is released. This is equivalent to the second filtering operation you performed using the menu bar tools.

   You can select **Edit | Undo** at any time to undo an edit. Select **File | Revert** to undo a series of edits. The Low-Pass Filter tool remains active until you either select another tool or click the Select tool.

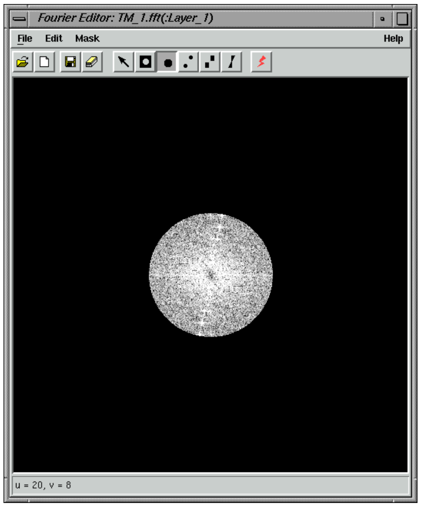
Use High-Pass Filtering

Next, you use the High-Pass Filter tool.

1. Select **Edit | Filter Options** from the Fourier Editor menu bar.

2. In the Filter Options dialog, click on the **Window Function** popup list and select **Hanning**.
3. Click **OK** in the Filter Options dialog.
4. Click the High-Pass Filter icon on the tool bar.
5. With your cursor in the center of the Fourier Editor, drag toward the right until the *u* coordinate in the status bar reads **20**. Release the mouse button.

   The image is filtered as soon as the mouse is released. The combination of filters is shown in the following example:



6. Select **File | Save As** from the menu bar.

   The Save Layer As dialog opens.

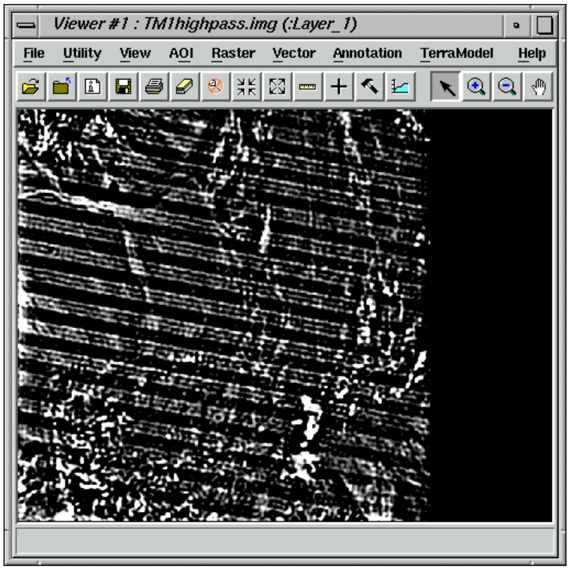
7. In the Save Layer As dialog, enter a new name for the .fft layer, such as **TM1highpass.fft.** Be sure to use a directory in which you have write permission.
8. Click **OK** to save the layer.
9. Click the Run icon on the tool bar or select **File | Inverse Transform** from the menu bar to create an inverse Fourier layer for display.
10. In the Inverse Fourier Transform dialog, enter a name for the new .img layer in the directory of your choice, such as **TM1highpass.img**.

11. Click **OK**.

    A Job Status dialog displays, indicating the progress of the function.

12. When the Job Status dialog indicates that the new .img layer is created, click **OK** and then display the layer in a Viewer.

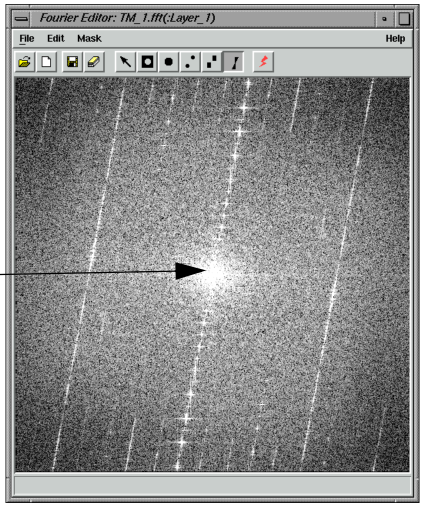
    Your new image should look similar to the following example:


Viewer #1 : TM1highpass.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Apply a Wedge Mask

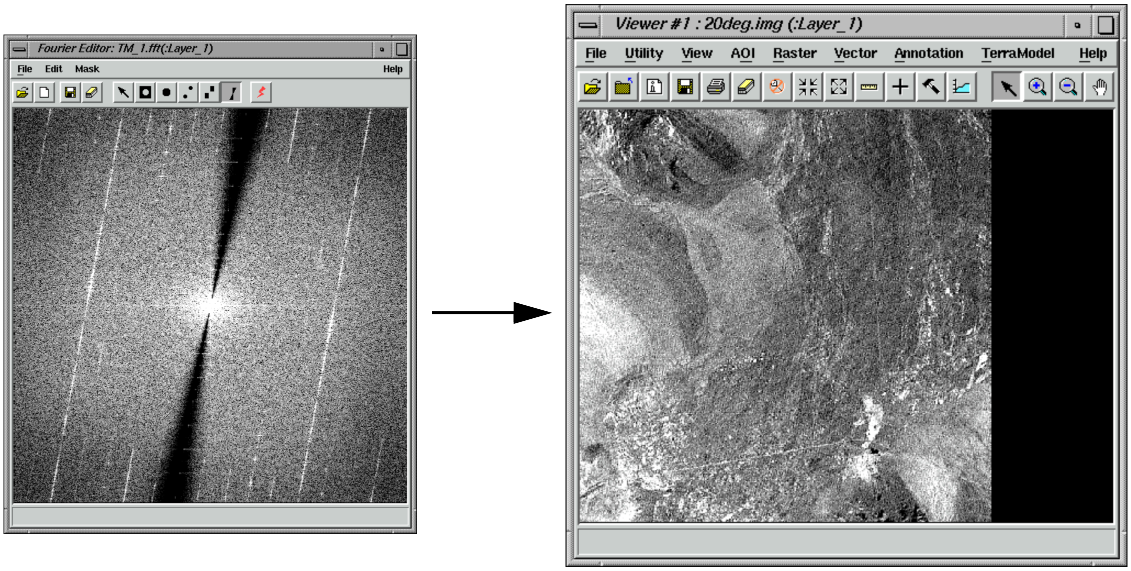
In the next exercise, you remove the nearly vertical radial line in the image, thereby removing the striping in the original image.

1. Redisplay the original .fft layer in the Fourier Editor if you have not already done so.
2. The Hanning window is still selected from the previous section, so you do not change it.
3. Click the Wedge Mask icon on the tool bar.
4. Using the following example as a guide, with your cursor over the center of the line, drag to the right until the lines of the wedge are about 20 degrees apart. Release the mouse button.


Fourier Editor: TM_1.fft(:Layer_1)
File
Edit
Mask
Help

Position your cursor in this area

The image is filtered as soon as you release the mouse, and similar to the following example. The resulting image is also shown.


Fourier Editor: TM_1.fft(:Layer_1)
File
Edit
Mask
Help
Viewer #1 : 20deg.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

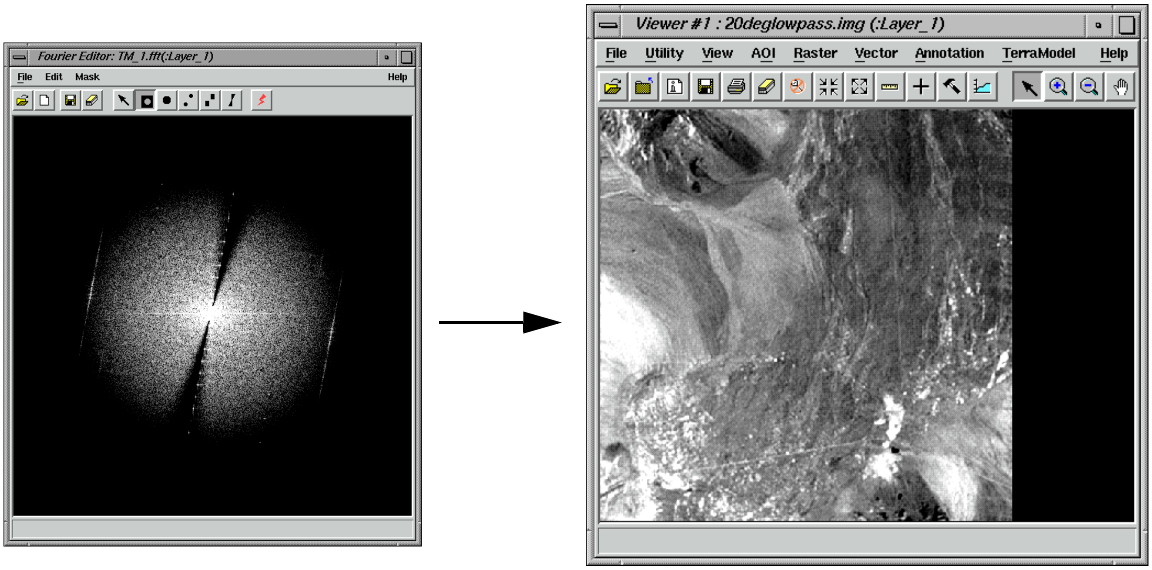
Combine Edits

You may combine as many edits as you like during an editing session. Since the Fourier Transform and Inverse Fourier Transform are linear operations, the effect of each edit on the resulting image is independent of the others. Here, you perform a low-pass filter over the wedged .fft layer that you just created.

1. With the .fft layer that you just created displayed in the Fourier Editor, click the Low-Pass Filter tool.

2. With your cursor in the center of the Fourier Editor, drag toward the right until the *u* coordinate in the status bar is about **200**.

   The .fft layer and resulting image look similar to the following:


Fourier Editor: TM_1.fft(:Layer_1)
File
Edit
Mask
Help
Viewer #1 : 20deglowpass.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

CHAPTER ELEVEN

Image Interpreter

Introduction

Image Interpreter is a group of over 50 functions that can be applied at the touch of a button to images with parameters you input. Most of the Image Interpreter functions are algorithms constructed as graphical models with Model Maker. These algorithms are common enhancements and utilities that have been made easily accessible through the Image Interpreter.

NOTE: Some of these functions are found in other parts of ERDAS IMAGINE, but are also listed in Image Interpreter for convenience.

The models used in Image Interpreter functions can be edited and adapted as needed with Model Maker (from Spatial Modeler) or the Spatial Modeler Language.

See the Spatial Modeler section of this manual for a description of the relationship between Spatial Modeler Language, Model Maker, and Image Interpreter. See "Chapter 11: Geographic Information Systems" *of the* ERDAS Field Guide *for more information on modeling.*

Approximate completion time for this tour guide is 50 minutes.

Subsetting an Image

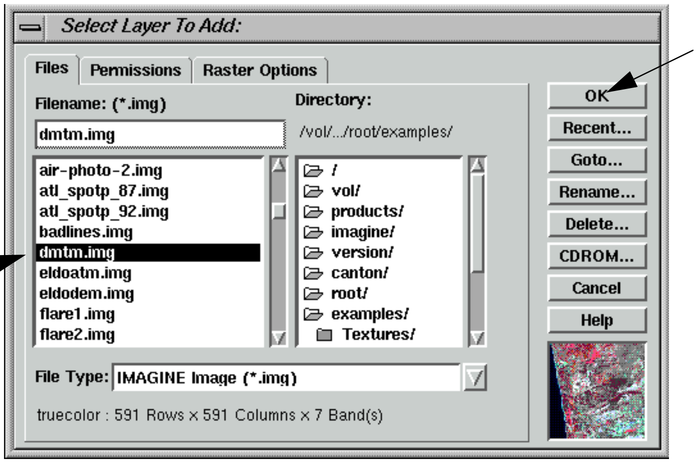
Many images used in IMAGINE cover a large area, while the actual area being studied can only cover a small portion of the image. To save on disk space and processing time, IMAGINE will let you make new images out of a subset of the entire dataset.

In this exercise, you will use the Subset Utility to take a subset of a small urbanized Area of Interest (AOI) from a much larger Landsat scene of San Diego.

ERDAS IMAGINE should be running with a Viewer open.

1. Select **File | Open Raster Layer** from the Viewer menu bar.

The Select Layer to add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
air-photo-2.img
atl_spotp_87.img
atl_spotp_92.img
badlines.img
dmtm.img
eldoatm.img
eldodem.img
flare1.img
flare2.img
/
vol/
products/
imagine/
version/
canton/
root/
examples/
Textures/

Filename: (*.img) dmtm.img

Directory: /vol/.../root/examples/

File Type: IMAGINE Image (*.img)

truecolor : 591 Rows x 591 Columns x 7 Band(s)

Click OK

Select dmtm.img

2. Select dmtm.img from the list of examples.

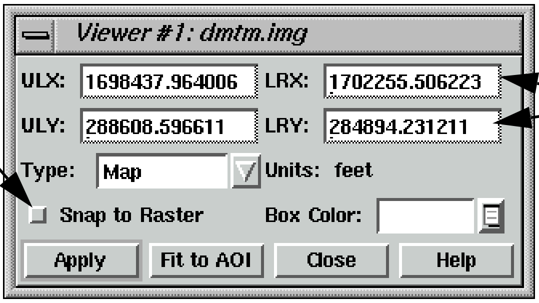
3. Click on **OK** to have the image display in the Viewer.

Selecting an AOI to Subset

In this section, you use the Subset utility to take a small Subset from a large image without using the Snap to Raster option.

1. With the cursor in the Viewer, right-click to access the Quick-View menu.

2. Select **Inquire Box...** from the Quick-View menu. The Inquire Box dialog displays. The title of this dialog is **Viewer #1: dmtm.img**.


Viewer #1: dmtm.img
Apply
Fit to AOI
Close
Help

ULX: 1698437.964006 LRX: 1702255.506223

ULY: 288608.596611 LRY: 284894.231211

Type: Map Units: feet

☐ Snap to Raster Box Color:

Uncheck the Snap to Raster checkbox

Enter these coordinates in the Inquire Box dialog.

3. Click on the Snap to Raster checkbox to uncheck this option. This will tell the Subset function to use the exact coordinates you enter for the Inquire Box.

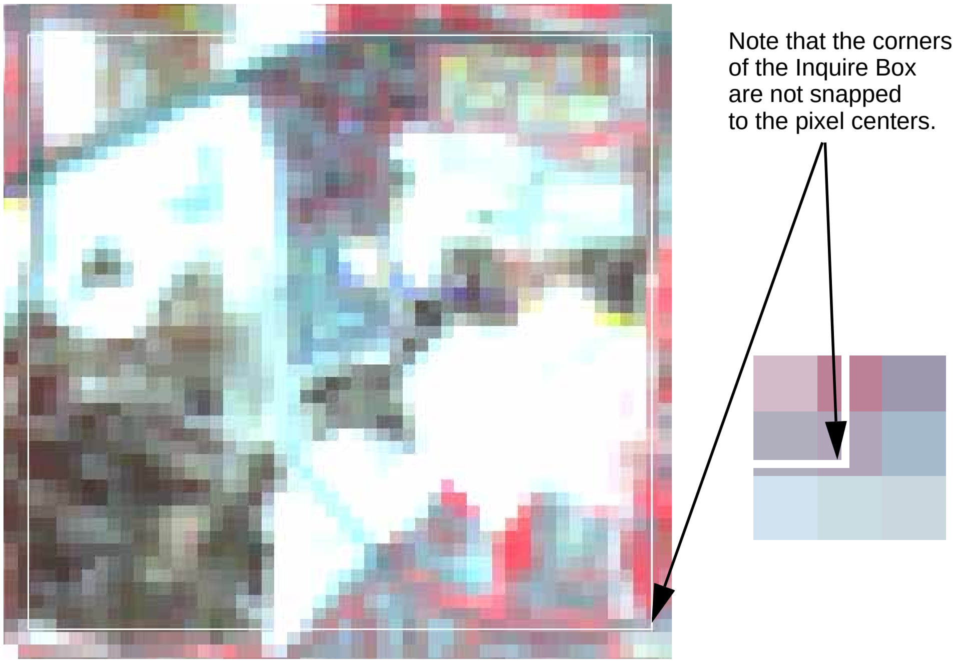
4. Enter the following coordinates into the Inquire Box dialog:

ULX: **1698385.570**
ULY: **288632.691217**
LRX: **1702282.557434**
LRY: **284900.708704**

5. Click **Apply** on the Inquire Box dialog. The Inquire Box moves to the new coordinates.

6. Use the **Zoom Tool** to zoom in on the Area of Interest.

The image in your Viewer should look something like this:


Note that the corners of the Inquire Box are not snapped to the pixel centers.

Subsetting an Image Without Snapping

1. Click on the Interpreter icon in the ERDAS IMAGINE icon panel.

The **Image Interpreter** menu opens. Each of the buttons in the **Image Interpreter** menu displays a submenu of Image Interpreter functions

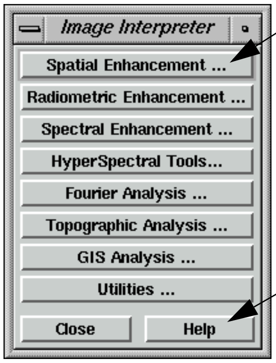
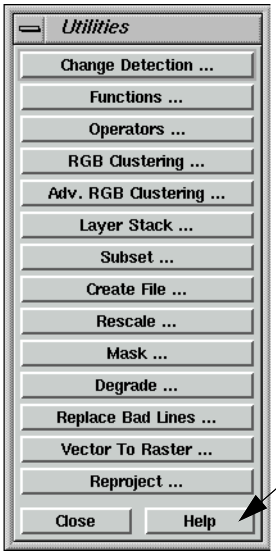


Image Interpreter
Spatial Enhancement ...
Radiometric Enhancement ...
Spectral Enhancement ...
HyperSpectral Tools...
Fourier Analysis ...
Topographic Analysis ...
GIS Analysis ...
Utilities ...
Close
Help

Ellipses indicate that these buttons displayanother dialog with additional options

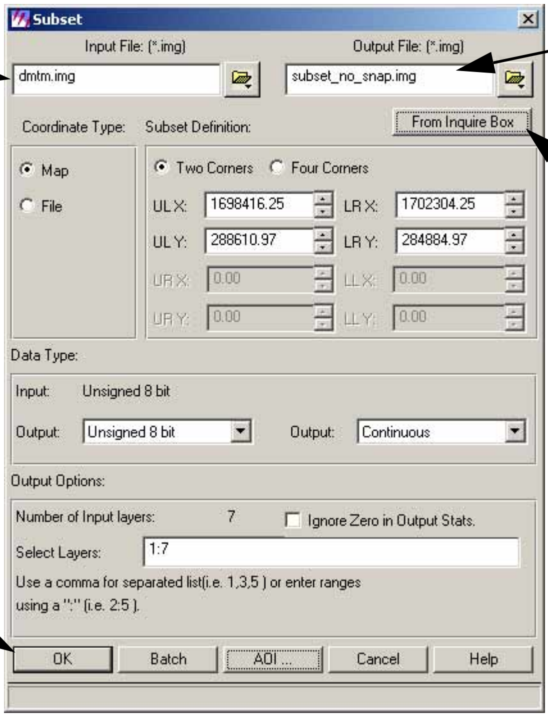
Click here to see an On-Line Help file describing this dialog

2. Select **Utilities** from the **Image Interpreter** menu and the **Utilities** menu opens


Utilities
Change Detection ...
Functions ...
Operators ...
RGB Clustering ...
Adv. RGB Clustering ...
Layer Stack ...
Subset ...
Create File ...
Rescale ...
Mask ...
Degrade ...
Replace Bad Lines ...
Vector To Raster ...
Reproject ...
Close
Help

Click here to see an On-Line Help file describing this dialog

3. Select **Subset** from the **Utilities** menu and the Subset dialog opens.


Subset
Coordinate Type:
Subset Definition:
From Inquire Box
Data Type:
Output Options:
OK
Batch
AOI ...
Cancel
Help

Input File: (*.img) dmtm.img

Output File: (*.img) subset_no_snap.img

Coordinate Type: ● Map ○ File

Subset Definition: ● Two Corners ○ Four Corners

UL X: 1698416.25 LR X: 1702304.25

UL Y: 288610.97 LR Y: 284884.97

UR X: 0.00 LL X: 0.00

UR Y: 0.00 LL Y: 0.00

Input: Unsigned 8 bit

Output: Unsigned 8 bit Output: Continuous

Number of Input layers: 7 ☐ Ignore Zero in Output Stats.

Select Layers: 1:7

Use a comma for separated list(i.e. 1,3,5) or enter ranges using a ":" (i.e. 2:5).

Enter input file here

Enter output file here

Click here to import coordinates from the Inquire Box in the Viewer

Click here to start the Subsetting process

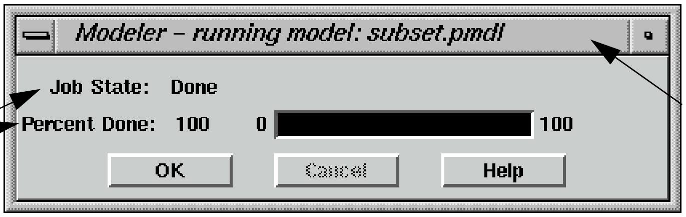
4. Under **Input File** in the Subset dialog, enter **dmtm.img**.

   This is a Landsat TM image of San Diego, California.

5. Under **Output File**, enter **subset_no_snap.img** in a directory where you have write permission.

6. Click **OK** to begin the Subsetting process.

   A Job Status bar displays, indicating the progress of the subsetting operation.


Modeler – running model: subset.pmdl
0
100
OK
Cancel
Help

Job State: Done

Percent Done: 100

Status updates here

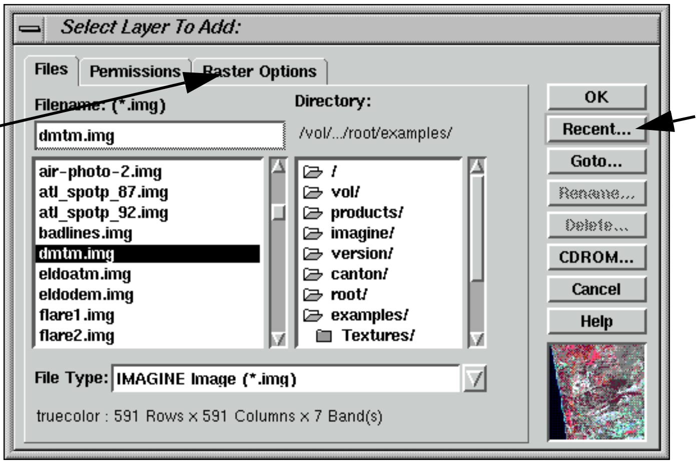
Title bar tells which function is running

7. Depending on your eml **Preferences** (under **Session | Preferences | User Interface & Session | Keep Job Status Box**), when the Job Status bar shows 100, indicating that the job is 100% done, you must either click **OK** to close the dialog or the dialog closes automatically.

Displaying the Subset with the Original Data

1. The original image should still be displayed in the Viewer.
2. Select **File | Open** from the menu bar on the Viewer.

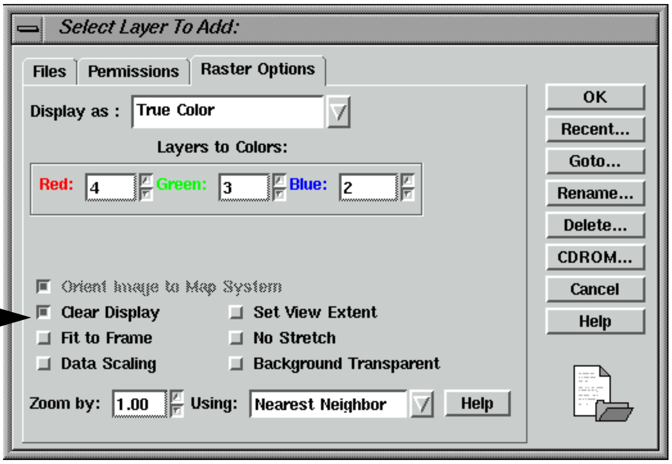
   The Select Layer to Add dialog displays.


Select Layer To Add:
Files
Permissions
Raster Options
Filename: (*.img)
dmtm.img
air-photo-2.img
atl_spotp_87.img
atl_spotp_92.img
badlines.img
dmtm.img
eldoatm.img
eldodem.img
flare1.img
flare2.img
Directory:
/vol/.../root/examples/
/
vol/
products/
imagine/
version/
canton/
root/
examples/
Textures/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
File Type: IMAGINE Image (*.img)
truecolor : 591 Rows x 591 Columns x 7 Band(s)

Click here to access the Raster Options tab

Click here to open a list of recently accessed files

3. Click the Recent button to open a list of recently accessed files.
4. Select **subset_no_snap.img** from the List of Recent Files.
5. Click **OK** to dismiss the List of Recent Files.
6. Click the Raster Options tab



Select Layer To Add:

Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
Layers to Colors:
Help

Display as : True Color

Red: 4 Green: 3 Blue: 2

☑ Orient Image to Map System

☑ Clear Display

☐ Set View Extent

☐ Fit to Frame

☐ No Stretch

☐ Data Scaling

☐ Background Transparent

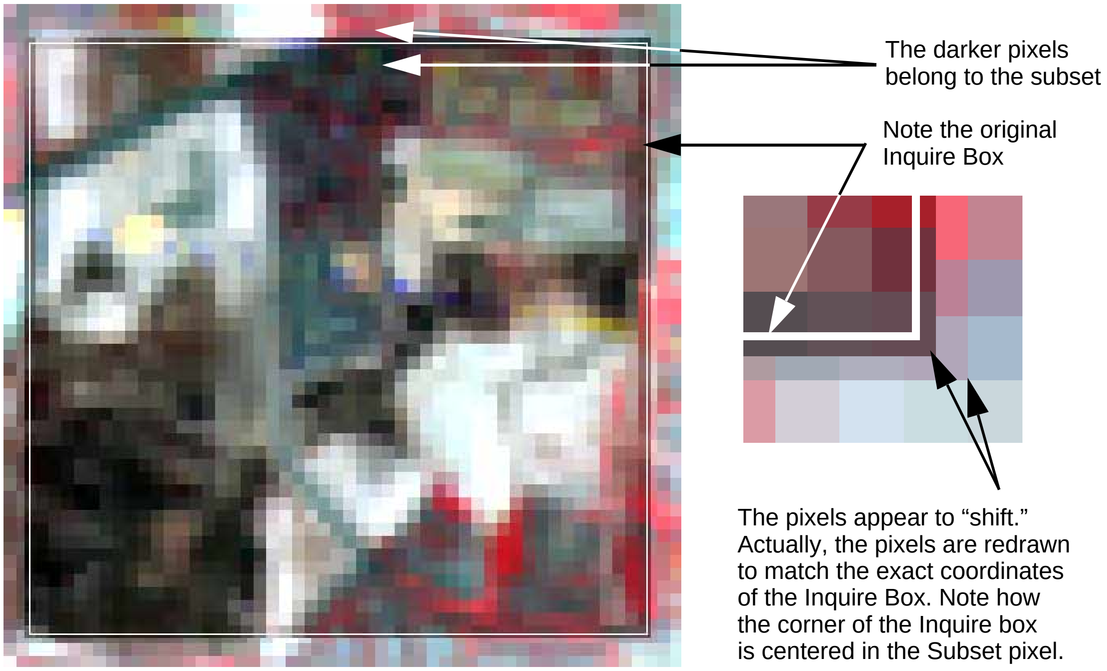
Zoom by: 1.00 Using: Nearest Neighbor

Click here to uncheck the Clear Display option

7. Uncheck the **Clear Display** option so the new subset of the original image will appear superimposed on the original image.

8. Click OK.

The subset displays in the Viewer over the original image.



The darker pixels belong to the subset

Note the original Inquire Box

The pixels appear to “shift.” Actually, the pixels are redrawn to match the exact coordinates of the Inquire Box. Note how the corner of the Inquire box is centered in the Subset pixel.

When a subset of an image is taken from an Inquire Box that is not snapped to the pixel grid of the original image, the subset is drawn using the exact coordinates of the of the Inquire Box. Because the subset pixel grid differs slightly from the original image's pixel grid, the subset image appears "shifted" from the original image.

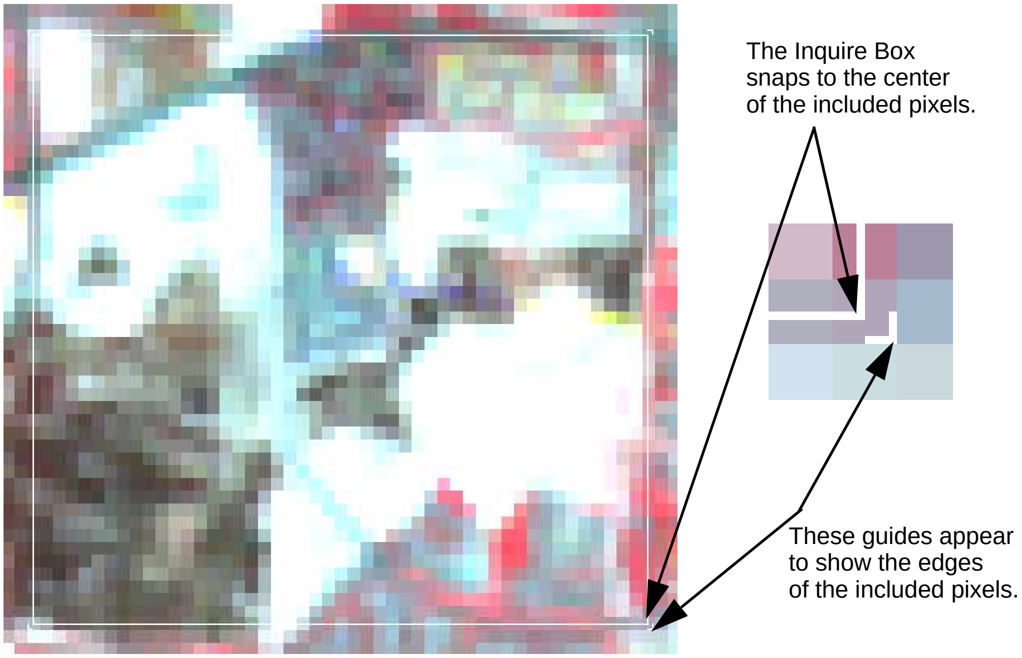
9. Click on the Clear Top Layer icon to remove **subset_no_snap.img** from the Viewer.

   The original image and the Inquire Box should still be displayed in the Viewer.

Subsetting an Image With Snap to Raster

1. In the Inquire Box dialog, click the **Snap to Raster** checkbox to make sure it is active (checked). Click **Apply**.

   The Inquire Box snaps to the pixels centers and looks like this:


The Inquire Box snaps to the center of the included pixels.
These guides appear to show the edges of the included pixels.

   The coordinates in the Inquire Box dialog update to reflect the new corner coordinates.

2. In the Utilities dialog, click on the **Subset** button. The Subset dialog displays.
3. Under **Input File**, enter **dmtm.img**.
4. In the **Output File**, enter **subset_snap.img** in a directory where you have write permission.
5. Click the **From Inquire Box** button.

The coordinates in the **Subset Definition** area update to reflect the corner coordinates of the Inquire Box.

6. Click **OK** to start the Subsetting Process.

   The Subsetting Progress meter appears, displaying the progress of the subsetting.

Viewing the Snapped Subset

1. Select **File | Open** from the menu bar in the Viewer.

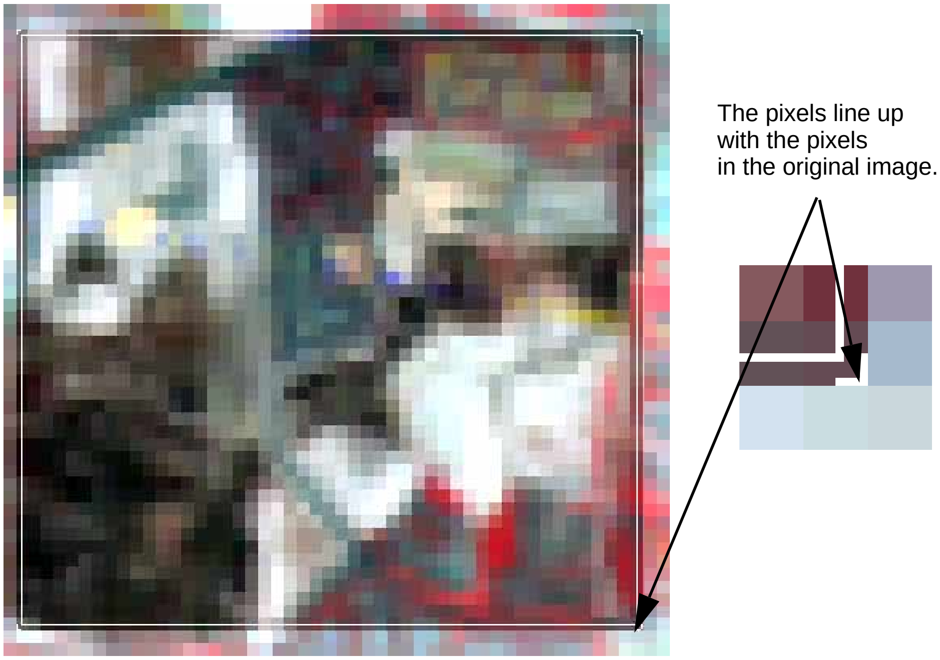
   The **Select Layer to Add** dialog opens.

2. Click on the Recent button.

   The **List of Recent Files** dialog displays.

3. Select **subset_snap.img** from the list.
4. Click **OK** on the List of Recent Files to close the dialog.
5. Click on the **Raster Options** tab in the Select Layer to Add dialog.
6. Deselect the **Clear Display** option.
7. Click **OK** to open **subset_snap.img** in the Viewer on top of dmtm.img.

   The subsetted image displays in the Viewer.



The pixels line up with the pixels in the original image.

Because the Inquire Box was snapped to the centers of the pixels before the image was processed, the pixels in the output file will line up exactly with the pixels in the original image.

8. Click **Close** on the Viewer.

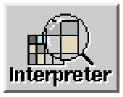
9. Click **Close** on the Utilities menu.

Apply Spatial Enhancement

ERDAS IMAGINE should be running with a Viewer open.

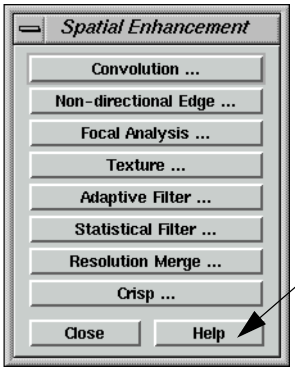
In this section, you use the convolution and crisp Spatial Enhancement functions to enhance images.

1. Click on the Interpreter icon in the ERDAS IMAGINE icon panel.


Interpreter

The **Image Interpreter** menu opens.

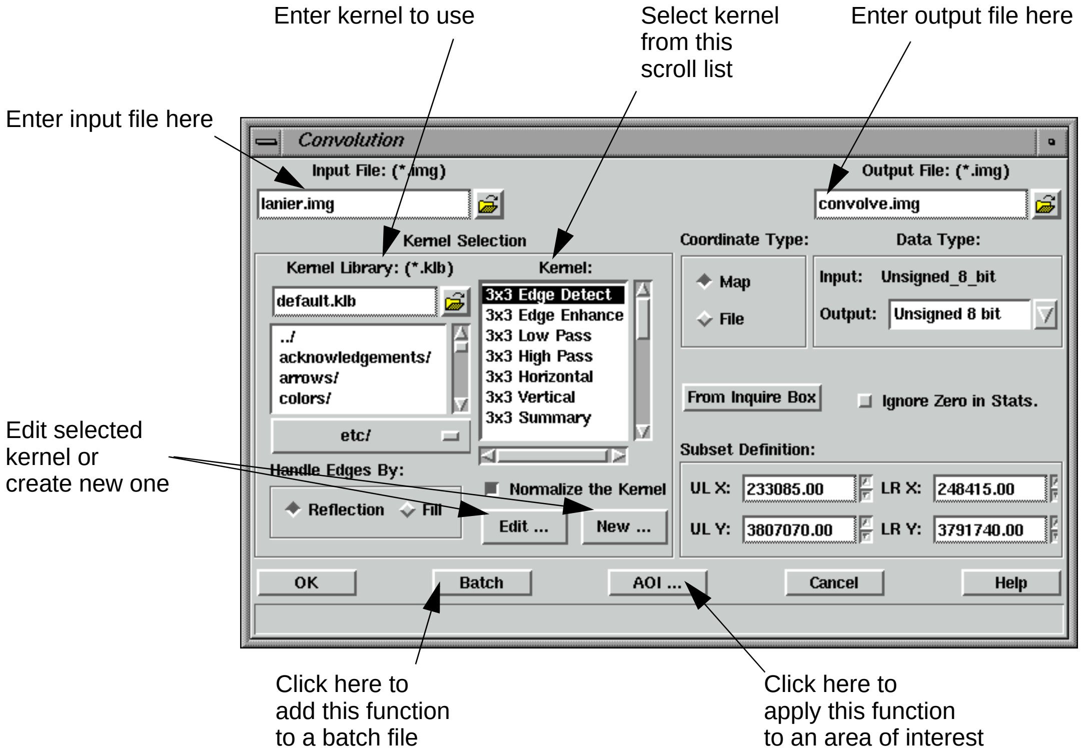
2. Select **Spatial Enhancement** from the **Image Interpreter** menu and the **Spatial Enhancement** menu opens.


Spatial Enhancement
Convolution ...
Non-directional Edge ...
Focal Analysis ...
Texture ...
Adaptive Filter ...
Statistical Filter ...
Resolution Merge ...
Crisp ...
Close
Help

Click here to see an On-Line Help file describing this dialog

Apply Convolution

1. Select **Convolution** from the **Spatial Enhancement** menu and the Convolution dialog opens.



Enter kernel to use

Select kernel from this scroll list

Enter output file here

Enter input file here

Edit selected kernel or create new one

Click here to add this function to a batch file

Click here to apply this function to an area of interest

Input File: (*.img) lanier.img

Output File: (*.img) convolve.img

Kernel Library: (*.klb) default.klb

Kernel: 3x3 Edge Detect

Coordinate Type: Map

Data Type: Input: Unsigned_8_bit

Output: Unsigned 8 bit

Handle Edges By: Reflection

Normalize the Kernel

UL X: 233085.00 LR X: 248415.00

UL Y: 3807070.00 LR Y: 3791740.00

Convolution
../
acknowledgements/
arrows/
colors/
etc/
Kernel Selection
3x3 Edge Detect
3x3 Edge Enhance
3x3 Low Pass
3x3 High Pass
3x3 Horizontal
3x3 Vertical
3x3 Summary
File
Fill
From Inquire Box
Ignore Zero in Stats.
Subset Definition:
Edit ...
New ...
OK
Batch
AOI ...
Cancel
Help

This interactive Convolution tool lets you perform convolution filtering on images. It provides a scrolling list of standard filters and lets you create new kernels. The new kernels can be saved to a library and used again at a later time.

NOTE: Do not close the **Image Interpreter** *menu, as you continue using it in the next section.*

Select Input/Output Files

1. In the Convolution dialog, under **Input File**, enter **lanier.img**.
2. Under **Output File**, enter **convolve.img** in the directory of your choice. It is not necessary to add the .img extension when typing the file name—ERDAS IMAGINE automatically appends the correct extension.

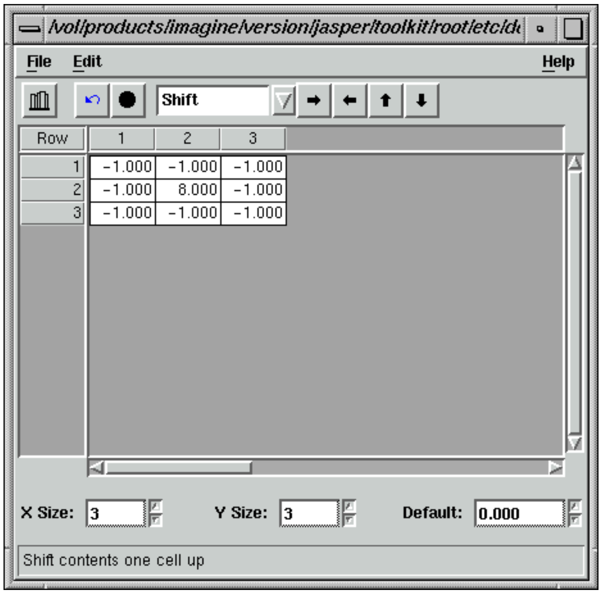
NOTE: Make sure you remember in which directory the output file is saved. This is important when you try to display the output file in a Viewer.

Select Kernel

Next, you must select the kernel to use for the convolution. A default kernel library containing some of the most common convolution filters is supplied with ERDAS IMAGINE. This library is opened in the **Kernel Selection** part of this dialog.

1. From the scrolling list under **Kernel**, click on **3x3 Edge Detect.**
2. Click on the **Edit** button in the **Kernel Selection** box.

The 3 × 3 Edge Detect dialog opens.



Row	1	2	3
1	-1.000	-1.000	-1.000
2	-1.000	8.000	-1.000
3	-1.000	-1.000	-1.000

X Size: 3

Y Size: 3

Default: 0.000

Shift contents one cell up

For this exercise, you use the Kernel Editor to simply view the kernel used for the 3 × 3 Edge Detect filter. However, if desired, you could make changes to the kernel at this time by editing the CellArray.

3. Select **File | Close** from the 3 × 3 Edge Detect dialog.

4. Click **OK** in the Convolution dialog.

   A Job Status dialog displays, indicating the progress of the function.

5. When the Job Status dialog shows that the process is 100% complete, click **OK**.

Check the File

1. Select **File | Open | Raster Layer** from the Viewer menu bar.

   The Select Layer To Add dialog opens

2. In the Select Layer To Add dialog under **Filename**, click on **lanier.img**.

3. Click **OK** to display the file in the Viewer.

4. Open a second Viewer window by clicking on the Viewer icon in the ERDAS IMAGINE icon panel.

5. Select **File | Open | Raster Layer** from the menu bar of the Viewer you just opened.

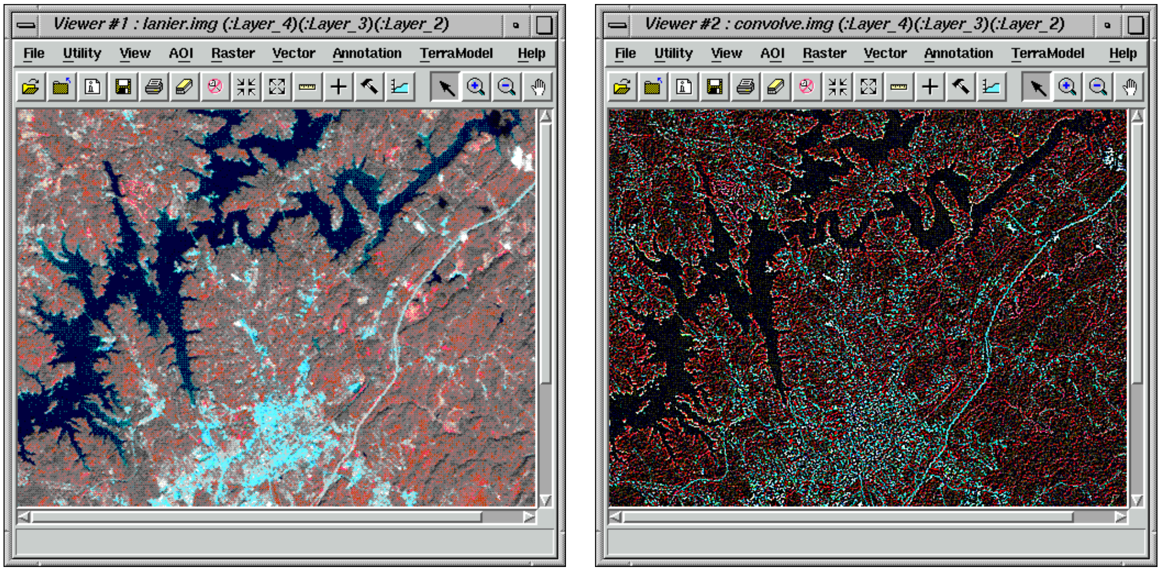
   The Select Layer To Add dialog opens.

6. In the Select Layer To Add dialog under **Filename**, enter the name of the directory in which you saved **convolve.img**, and press the Return key on your keyboard.

7. In the list of files, click on **convolve.img** and then click **OK**.

   The output file generated by the Convolve function, **convolve.img**, displays in the second Viewer.

8. In the ERDAS IMAGINE menu bar, select **Session | Tile Viewers** to compare the two files side by side.

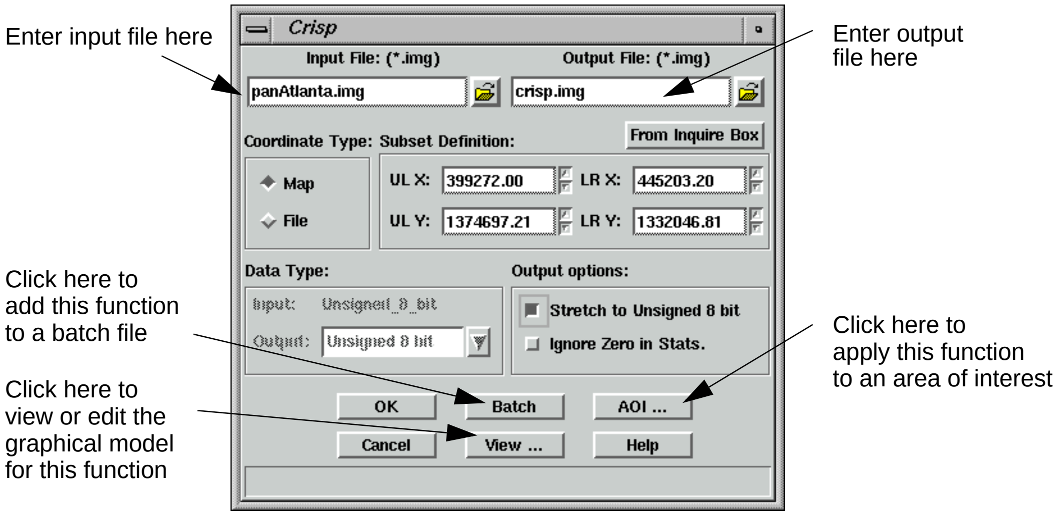

Viewer #1 : lanier.img (:Layer_4)(:Layer_3)(:Layer_2)
File Utility View AOI Raster Vector Annotation TerraModel Help
Viewer #2 : convolve.img (:Layer_4)(:Layer_3)(:Layer_2)
File Utility View AOI Raster Vector Annotation TerraModel Help

9. When you are finished comparing the two files, select **File | Clear** from the menu bar of each Viewer.

Apply Crisp

1. Select **Crisp** from the **Spatial Enhancement** menu.

   The Crisp dialog opens.



Input File: (*.img) panAtlanta.img

Output File: (*.img) crisp.img

Coordinate Type: ◆ Map ◇ File

Subset Definition: UL X: 399272.00 LR X: 445203.20 UL Y: 1374697.21 LR Y: 1332046.81 From Inquire Box

Data Type: Input: Unsigned 8 bit Output: Unsigned 8 bit

Output options: ☑ Stretch to Unsigned 8 bit ☐ Ignore Zero in Stats.

Enter input file here

Enter output file here

Click here to add this function to a batch file

Click here to view or edit the graphical model for this function

Click here to apply this function to an area of interest

The Crisp dialog is a good example of the basic Image Interpreter dialog. Other dialogs may have more prompts for inputs, depending on the function. Each dialog opens with default entries that are acceptable for use. These entries can be changed, if necessary, to achieve specific results.

2. Under **Input File** in the Crisp dialog, enter **panAtlanta.img**.

   This is a SPOT panchromatic scene of downtown Atlanta, Georgia.

3. Under **Output File**, enter **crisp.img** in the directory of your choice as the output file.

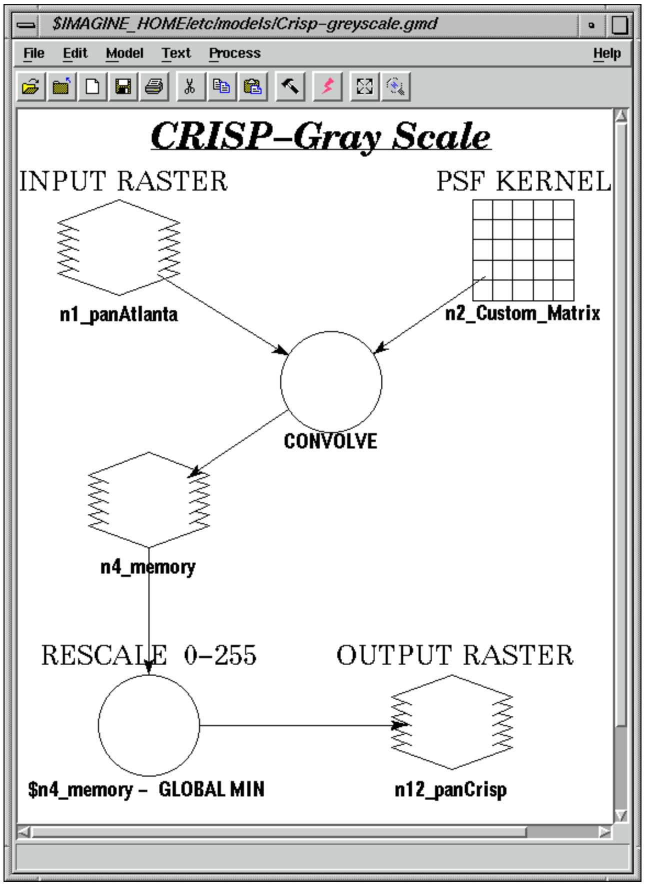
4. Under **Output Options** in the Crisp dialog, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

   This option produces the output file in unsigned 8-bit format, which saves disk space.

Use the View Option

1. Click on the **View** button at the bottom of the Crisp dialog.

   The Model Maker viewer window opens and displays the graphical model used for the Crisp function.


$IMAGINE_HOME/etc/models/Crisp-greyscale.gmd
File
Edit
Model
Text
Process
Help
CRISP–Gray Scale
INPUT RASTER
PSF KERNEL
n1_panAtlanta
n2_Custom_Matrix
CONVOLVE
n4_memory
RESCALE 0–255
OUTPUT RASTER
$n4_memory - GLOBAL MIN
n12_panCrisp

The **View** button in each Image Interpreter dialog lets you view the graphical model behind each function. If you want to change the model for a specific purpose, you can edit it through the Model Maker and apply the edited function to the image by running the model in Model Maker.

See "Chapter 16: Spatial Modeler" *for information on editing and running a model in Model Maker.*

2. Exit the Model Maker by selecting **File | Close All**.

3. Click **OK** in the Crisp dialog to start the process.

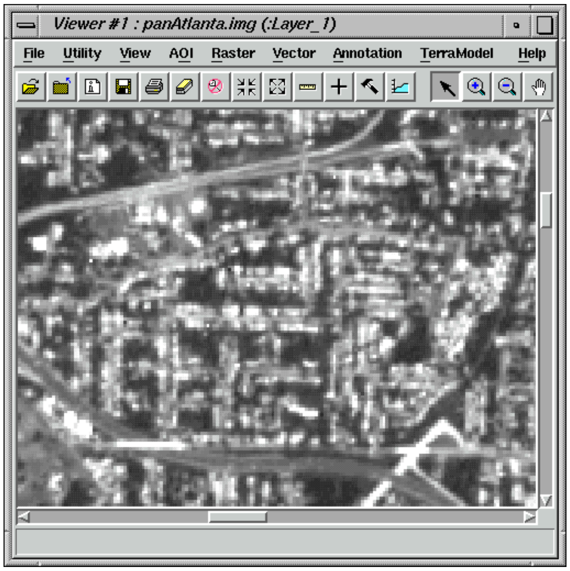
   A Job Status dialog opens, indicating the progress of the function.

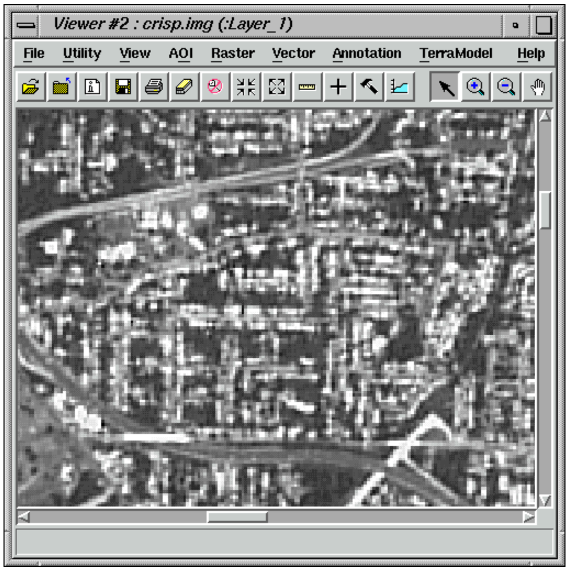
4. When the Job Status dialog shows that the process is 100% complete, click **OK**.

View Results

1. Display **panAtlanta.img** in a Viewer.

2. Display **crisp.img**, the output file generated by the Crisp function, in the other Viewer.
3. Note the differences between the two images; **crisp.img** appears to be sharper.


Viewer #1 : panAtlanta.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help


Viewer #2 : crisp.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

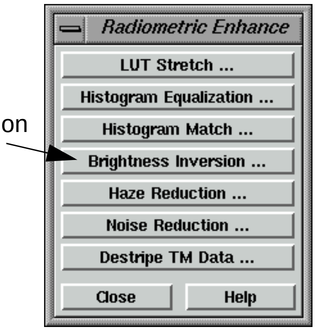
4. Use the Zoom In tool in the Viewer tool bar to zoom in for a closer look at the crispening of the image in **crisp.img**.
5. When you are through, close all the Viewers at once by selecting **Session | Close All Viewers** from the ERDAS IMAGINE menu bar.
6. Click **Close** in the **Spatial Enhancement** menu.

NOTE: Do not close the **Image Interpreter** *menu, as you continue using it in the next section.*

Apply Radiometric Enhancement

1. In the **Image Interpreter** menu, select **Radiometric Enhancement**.

The **Radiometric Enhance** menu opens.


Radiometric Enhance
LUT Stretch ...
Histogram Equalization ...
Histogram Match ...
Brightness Inversion ...
Haze Reduction ...
Noise Reduction ...
Destripe TM Data ...
Close
Help

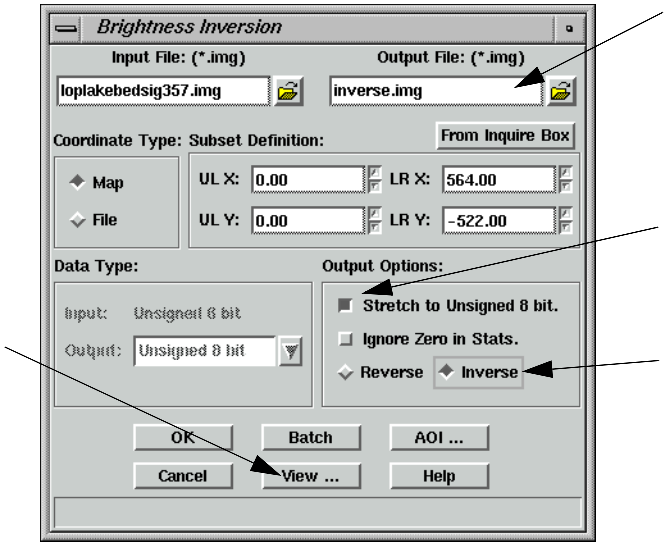
Click here to access the Brightness Inversion function

In this section, you use both the **Inverse** and **Reverse** options of the **Image Inversion** function to enhance images. Inverse emphasizes detail in the dark portions of an image. Reverse simply reverses the DN values.

Apply Brightness Inversion

1. In the **Radiometric Enhance** menu, select **Brightness Inversion**.

   The Brightness Inversion dialog opens.


Brightness Inversion
Coordinate Type:
Subset Definition:
From Inquire Box
Data Type:
Output Options:
OK
Batch
AOI ...
Cancel
View ...
Help

Input File: (*.img) loplakebedsig357.img

Output File: (*.img) inverse.img

◆ Map ◇ File

UL X: 0.00 LR X: 564.00

UL Y: 0.00 LR Y: -522.00

Input: Unsigned 8 bit

Output: Unsigned 8 bit

☒ Stretch to Unsigned 8 bit.

☐ Ignore Zero in Stats.

◇ Reverse ◆ Inverse

Enter output file name here

Select this option to reduce file size

Click here to see model in Model Maker

Select option here

2. In the Brightness Inversion dialog under **Input File**, enter **loplakebedsig357.img**.
3. Under **Output File**, enter **inverse.img** in the directory of your choice.
4. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.
5. Under **Output Options**, click on **Inverse**.

6. Click **OK** in the Brightness Inversion dialog to start the process.

   A Job Status dialog displays, indicating the progress of the function.

Reverse

1. Select **Brightness Inversion** from the **Radiometric Enhance** menu.

   The Brightness Inversion dialog opens.

2. In the Brightness Inversion dialog, enter **loplakebedsig357.img** as the input file.
3. Enter **reverse.img** in the directory of your choice as the **Output File**.
4. Turn on the **Stretch to Unsigned 8 bit** checkbox under **Output Options**.
5. Click **OK** in the Brightness Inversion dialog to start the process.

   A Job Status dialog displays, indicating the progress of the function.

View Changes

1. Open a Viewer and display **inverse.img**.
2. Right-hold within the Viewer and select **Fit Window to Image** from the **Quick View** menu.

   The Viewer changes size to bound the image data.

3. Select **View | Split | Split Vertical** from the Viewer menu bar.

   A second Viewer opens.

4. In the second Viewer, click on the Open icon (this is the same as selecting **File | Open | Raster Layer** from the Viewer menu bar).

   The Select Layer To Add dialog opens.

5. From the Select Layer To Add dialog, open the file **reverse.img**.
6. In the second Viewer, select **View | Split | Split Vertical** from the Viewer menu bar.

   A third Viewer opens.

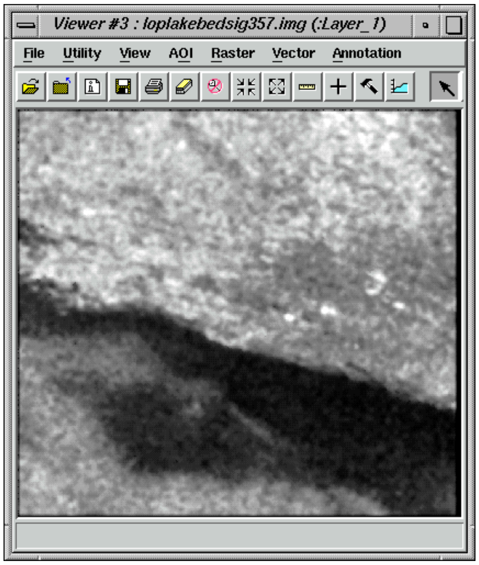
7. With your cursor in Viewer #3, press **Ctrl-r** on your keyboard (this is just another way to open a raster layer).

   The Select Layer To Add dialog opens.

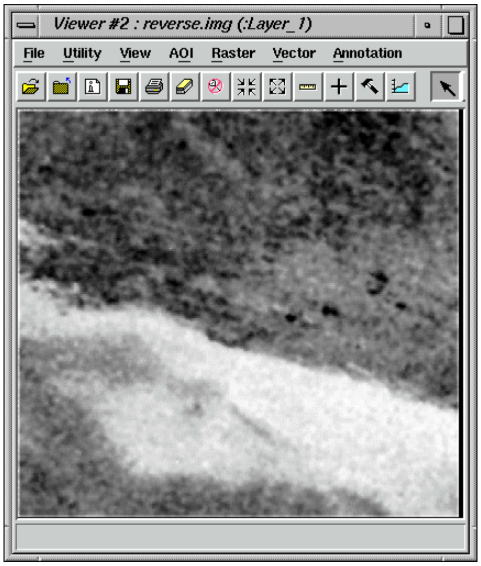
8. From the Select Layer To Add dialog, open the file **loplakebedsig357.img**.

9. Resize the Viewers on screen so that you can see all three Viewers. Note the differences between **reverse.img** and **inverse.img** compared to the original file.

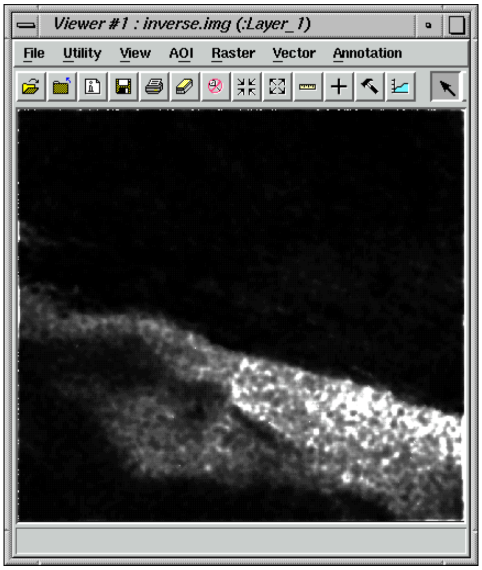
Original image


Viewer #3 : loplakebedsig357.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation

Reverse image


Viewer #2 : reverse.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation

Inverse image


Viewer #1 : inverse.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation

10. When you are through, close all three Viewers by selecting **Session | Close All Viewers** from the ERDAS IMAGINE menu bar.

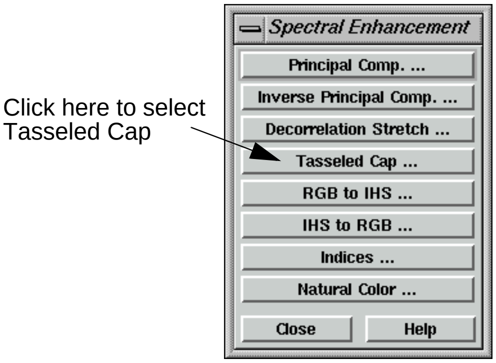
11. Click **Close** in the **Radiometric Enhance** menu.

NOTE: Do not close the **Image Interpreter** *menu, as you continue using it in the next section.*

Apply Spectral Enhancement

1. In the **Image Interpreter** menu, click on **Spectral Enhancement**.

   The **Spectral Enhancement** menu opens.


Spectral Enhancement
Principal Comp. ...
Inverse Principal Comp. ...
Decorrelation Stretch ...
Tasseled Cap ...
RGB to IHS ...
IHS to RGB ...
Indices ...
Natural Color ...
Close
Help

Click here to select Tasseled Cap

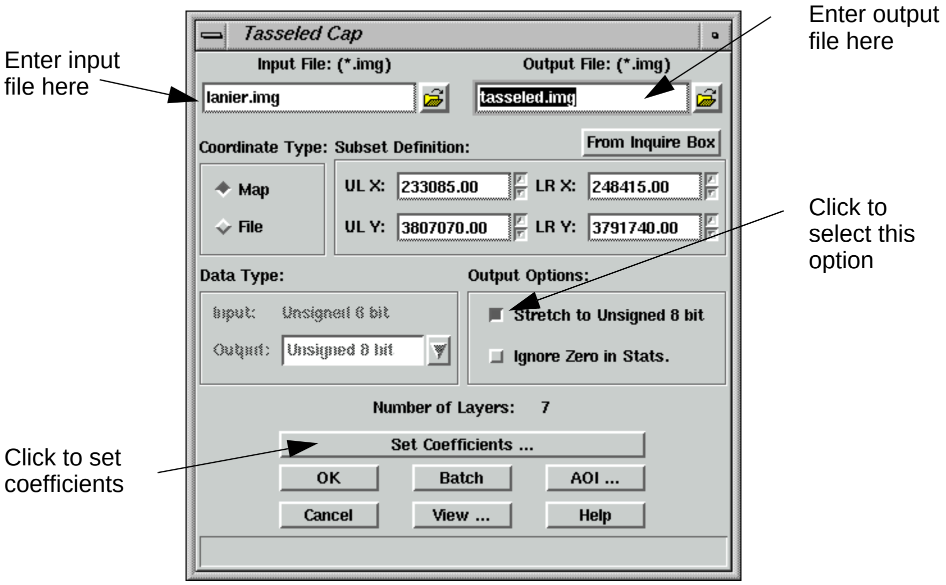
In this section, you use the following Spectral Enhancement functions:

- Tasseled Cap
- RGB to IHS
- IHS to RGB
- Indices

Use Tasseled Cap

1. In the **Spectral Enhancement** menu, select **Tasseled Cap**.

   The Tasseled Cap dialog opens.


Tasseled Cap
Input File: (*.img)
Output File: (*.img)
Coordinate Type:
Subset Definition:
From Inquire Box
Map
File
Data Type:
Output Options:
OK
Batch
AOI ...
Cancel
View ...
Help

Enter input file here

Enter output file here

Click to select this option

Click to set coefficients

Input File: lanier.img

Output File: tasseled.img

Coordinate Type: ● Map ○ File

Subset Definition:

UL X: 233085.00
LR X: 248415.00
UL Y: 3807070.00
LR Y: 3791740.00

Input: Unsigned 8 bit

Output: Unsigned 8 bit

☑ Stretch to Unsigned 8 bit

☐ Ignore Zero in Stats.

Number of Layers: 7

Set Coefficients ...

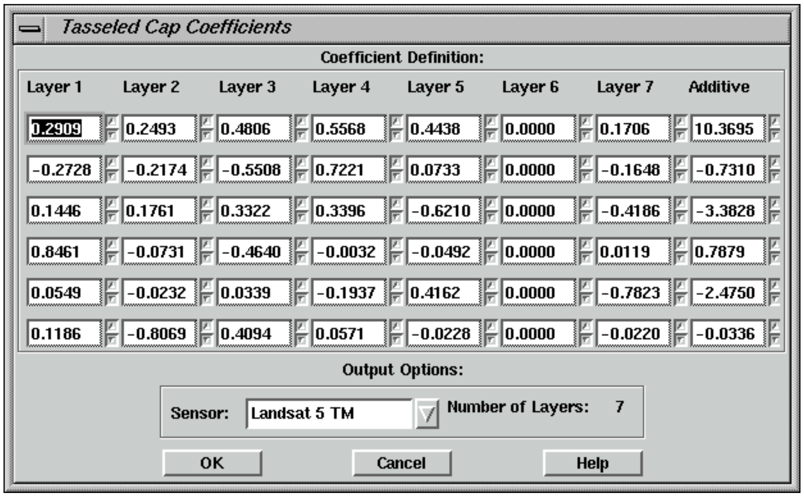
2. Under **Input File**, enter **lanier.img**. That image is a Landsat TM image of Lake Lanier, Georgia, which was obtained by the Landsat 5 sensor.

3. Enter **tasseled.img** in the directory of your choice as the **Output File** name.

4. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

5. Click on **Set Coefficients**.

   The Tasseled Cap Coefficients dialog opens.


Tasseled Cap Coefficients
OK
Cancel
Help

Coefficient Definition:

Layer 1	Layer 2	Layer 3	Layer 4	Layer 5	Layer 6	Layer 7	Additive
0.2909	0.2493	0.4806	0.5568	0.4438	0.0000	0.1706	10.3695
-0.2728	-0.2174	-0.5508	0.7221	0.0733	0.0000	-0.1648	-0.7310
0.1446	0.1761	0.3322	0.3396	-0.6210	0.0000	-0.4186	-3.3828
0.8461	-0.0731	-0.4640	-0.0032	-0.0492	0.0000	0.0119	0.7879
0.0549	-0.0232	0.0339	-0.1937	0.4162	0.0000	-0.7823	-2.4750
0.1186	-0.8069	0.4094	0.0571	-0.0228	0.0000	-0.0220	-0.0336

Output Options:

Sensor: Landsat 5 TM

Number of Layers: 7

The coefficients that display are the standard default entries for Landsat 5 TM Tasseled Cap transformation. For this exercise, you use the default entries, although you may change these entries at any time.

6. Click **OK** in the Tasseled Cap Coefficients dialog.

7. Click **OK** in the Tasseled Cap dialog to start the function.

   A Job Status dialog opens to report the state of the job.

8. When the Job Status dialog indicates that the job is **Done**, click **OK**.

Check Results

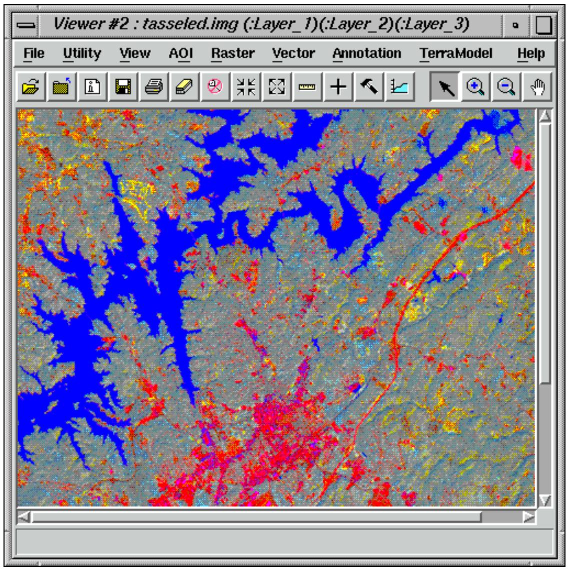
1. Open a Viewer and display **lanier.img**.

2. Open a second Viewer and then open the Select Layer To Add dialog by clicking on the Open icon in the Viewer tool bar.

3. In the Select Layer To Add dialog, enter the name of the directory in which you saved **tasseled.img**, press Return on your keyboard, and then click **tasseled.img** in the file list to select it.

4. Click the **Raster Options** tab at the top of the Select Layer To Add dialog. Under **Layers to Colors**, use layer **1** as **Red**, layer **2** as **Green**, and layer **3** as **Blue**.

5. Click **OK** in the Select Layer To Add dialog.



The image, **tasseled.img**, shows a degree of brightness, greenness, and wetness, as calculated by the Tasseled Cap coefficients used.

- Layer 1 (red) = the brightness component (indicates areas of low vegetation and high reflectors)
- Layer 2 (green) = the greenness component (indicates vegetation)
- Layer 3 (blue) = the wetness component (indicates water or moisture)

6. When you are through, close the Viewers by selecting **Session | Close All Viewers** from the ERDAS IMAGINE menu bar.

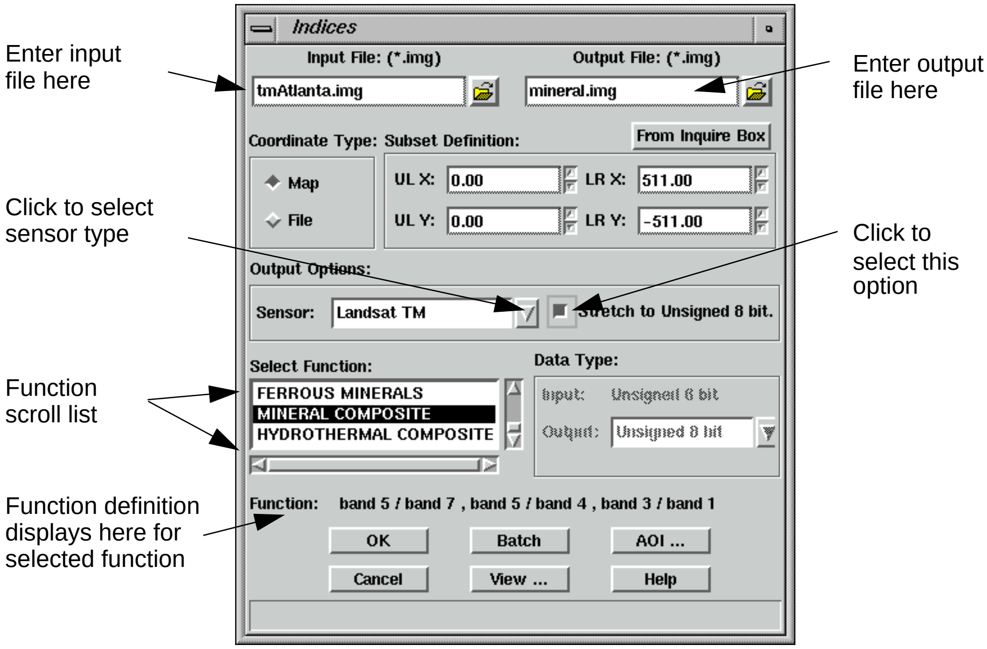
Use the Indices Function

Next, you apply a mineral ratio from the **Indices** function to a Landsat TM image. Such ratios are commonly used by geologists searching for specific mineral deposits in the earth.

For more information on this transformation, see "Chapter 5: Enhancement" *in the* ERDAS Field Guide.

1. In the **Spectral Enhancement** menu, select **Indices**.

The Indices dialog opens.



Indices

Input File: (*.img) tmAtlanta.img

Output File: (*.img) mineral.img

Coordinate Type: ◆ Map ◇ File

Subset Definition:

UL X: 0.00
LR X: 511.00
UL Y: 0.00
LR Y: -511.00

Output Options:

Sensor: Landsat TM

☑ Stretch to Unsigned 8 bit.

Select Function:

FERROUS MINERALS
MINERAL COMPOSITE
HYDROTHERMAL COMPOSITE

Data Type:

Input: Unsigned 8 bit
Output: Unsigned 8 bit

Function: band 5 / band 7 , band 5 / band 4 , band 3 / band 1

Enter input file here
Click to select sensor type
Function scroll list
Function definition displays here for selected function
Enter output file here
Click to select this option
From Inquire Box
OK
Batch
AOI ...
Cancel
View ...
Help

2. Under **Input File**, enter **tmAtlanta.img**.

3. Enter **mineral.img** in the directory of your choice as the **Output File**.

4. Under **Select Function**, click **MINERAL COMPOSITE** in the scrolling list.

This index is a composite of three mineral ratios.

- Clay minerals = band 5 / band 7
- Ferrous minerals = band 5 / band 4
- Iron oxide = band 3 / band 1

NOTE: Notice how the selected function is defined beside the **Function** *label, underneath the* **Select Function** *scroll list.*

5. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

6. Click **OK** in the Indices dialog to start the process.

   A Job Status dialog displays, indicating the progress of the function.

7. When the Job Status dialog indicates that the job is **Done**, click **OK**.

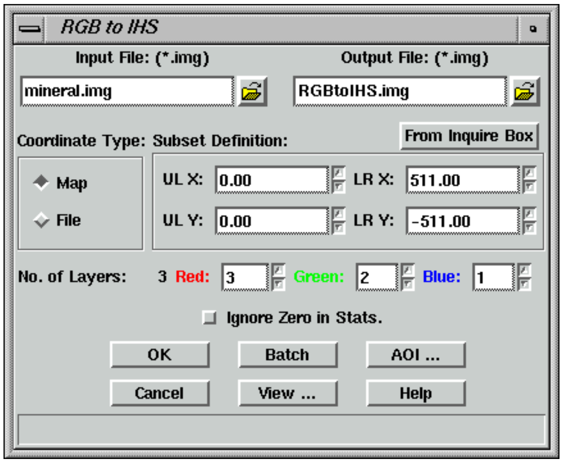
Choose RGB to IHS

Next, you use the **RGB to IHS** function (red, green, blue to intensity, hue, saturation) and the reverse **IHS to RGB** function to enhance the image information obtained by this mineral ratio.

The purpose of this function is to produce an input file for the IHS to RGB function.

1. Select **RGB to IHS** from the **Spectral Enhancement** menu.

   The **RGB to IHS** dialog opens.



RGB to IHS

Input File: (*.img) mineral.img

Output File: (*.img) RGBtoIHS.img

Coordinate Type: ● Map ○ File

Subset Definition:

UL X: 0.00

UL Y: 0.00

LR X: 511.00

LR Y: -511.00

No. of Layers: 3 Red: 3 Green: 2 Blue: 1

☐ Ignore Zero in Stats.

2. Enter the mineral ratio output from the previous exercise (**mineral.img**) as the **Input File**.

3. Enter **RGBtoIHS.img** (in the directory of your choice) as the **Output File**.

4. Click **OK** in the RGB to IHS dialog.

   A Job Status dialog displays, reporting the progress of the function.

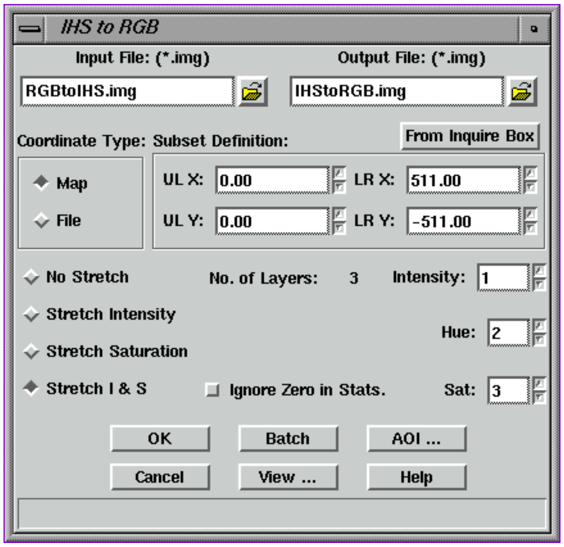
5. When the Job Status dialog indicates that the job is **Done**, click **OK**.

Choose IHS to RGB

Now, you convert the IHS image back into an RGB image.

1. Select **IHS to RGB** from the **Spectral Enhancement** menu.

   The IHS to RGB dialog opens.



IHS to RGB

Input File: (*.img) RGBtoIHS.img

Output File: (*.img) IHStoRGB.img

Coordinate Type: ◆ Map ◇ File

Subset Definition: UL X: 0.00 LR X: 511.00 UL Y: 0.00 LR Y: -511.00

◇ No Stretch
◇ Stretch Intensity
◇ Stretch Saturation
◆ Stretch I & S

No. of Layers: 3

Intensity: 1
Hue: 2
Sat: 3

☐ Ignore Zero in Stats.

2. In the IHS to RGB dialog, enter **RGBtoIHS.img** output from the previous exercise as the **Input File**.

3. Enter **IHStoRGB.img** as the **Output File** in the directory of your choice.

4. On the IHS to RGB dialog, click **Stretch I & S**.

   This option applies a global Min-Max contrast stretch to the Intensity and Saturation values in the image before converting.

5. Click **OK** in the IHS to RGB dialog.

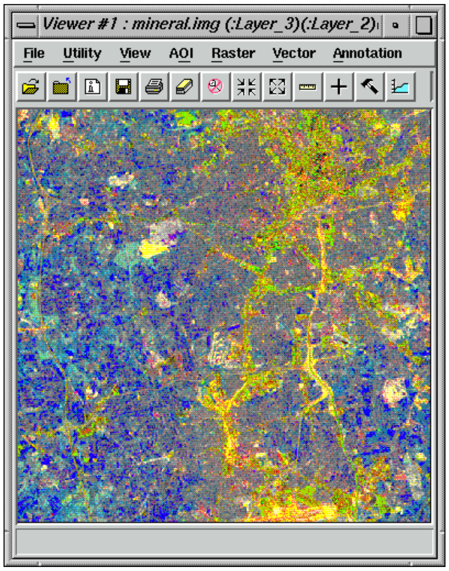
   A Job Status dialog displays, reporting the progress of the function.

6. When the Job Status dialog indicates that the job is **Done**, click **OK**.

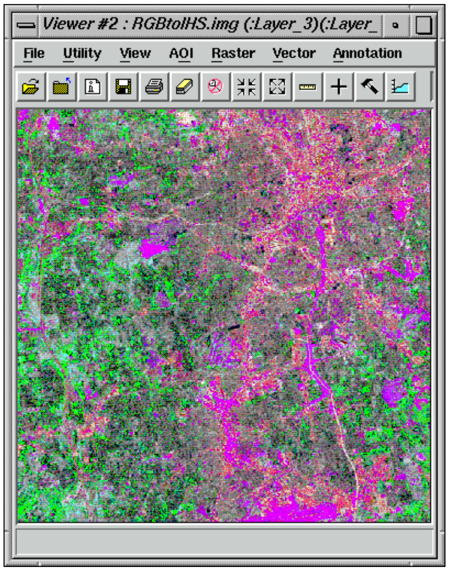
View the Results

1. Open three Viewers and then open the following files for comparison.

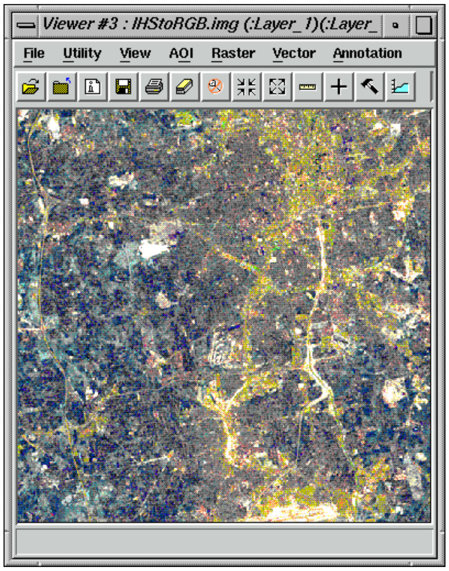
   - **mineral.img**—mineral ratio index. Proper interpretation can reveal the presence or absence of iron, clay, or ferrous minerals.


Viewer #1 : mineral.img (:Layer_3)(:Layer_2)
File
Utility
View
AOI
Raster
Vector
Annotation

- **RGBtoIHS.img**—red, green, and blue values converted to intensity, hue, and saturation values. This image does not appear similar to the input file. It is not meant for interpretation; it is only meant to produce an input for the IHS to RGB function.


Viewer #2 : RGBtoIHS.img (:Layer_3)(:Layer_
File
Utility
View
AOI
Raster
Vector
Annotation

- **IHStoRGB.img** (see special instructions below)—intensity, hue, and saturation values converted to red, green, and blue values (appears similar to **mineral.img**). The intensity and saturation (red and blue) values have been contrast-stretched for better interpretation.


Viewer #3 : IHStoRGB.img (:Layer_1)(:Layer_
File
Utility
View
AOI
Raster
Vector
Annotation

In the Open Raster Layer dialog, when displaying **IHStoRGB.img***, be sure to load Layer 1 as Red, Layer 2 as Green, and Layer 3 as Blue. This is because the order of the layers was reversed in the transformation.*

2. When you are through comparing the files, close the Viewers by selecting **Session | Close All Viewers** in the ERDAS IMAGINE menu bar.

3. Click **Close** in the **Image Interpreter** menu.

   The **Image Interpreter** and **Spectral Enhancement** menus close.

CHAPTER TWELVE

Orthorectification

Introduction

Rectification is the process of projecting the data onto a plane and making it conform to a map projection system. Assigning map coordinates to the image data is called georeferencing. Since all map projection systems are associated with map coordinates, rectification involves georeferencing.

The orthorectification process removes the geometric distortion inherent in imagery caused by camera/sensor orientation, topographic relief displacement, and systematic errors associated with imagery. Orthorectified images are planimetrically true images that represent ground objects in their true “real world” X and Y positions. For these reasons, orthorectified imagery has become accepted as the ideal reference image backdrop necessary for the creation and maintenance of vector data contained within a GIS.

By performing space resection, the effects of camera/sensor orientation have been considered and removed. By defining a DEM or constant elevation value (ideal for use in areas containing minimal relief variation), the effects of topographic relief displacement can be considered and removed.

For information on block bundle adjustment, see the IMAGINE OrthoBASE™ Tour Guide and the IMAGINE OrthoBASE™ User’s Manual.

Approximate completion time for this tour guide is 30 minutes.

Rectify a Camera Image

In this tour guide, you orthorectify a camera image of Palm Springs, California, using a NAPP (National Aerial Photography Program) photo.

Perform Image to Image Rectification

In rectifying the camera image, you use these basic steps:

- display Camera image
- start Geometric Correction Tool
- enter the Camera model properties
- record GCPs
- resample or calibrate the image

ERDAS IMAGINE should be running and a Viewer open. You must have write permission in a file if you wish to calibrate it. In a command shell, copy **ps_napp.img** to a directory in which you have write permission and at least 10 Mb of space. Set the permissions on **ps_napp.img** to read, write, execute using the `chmod 777 ps_napp.img` command (on UNIX).

Next, you should verify that **ps_napp.img** has no map or projection information. Select **Tools | Image Information** from the ERDAS IMAGINE menu bar. Open your copy of **ps_napp.img**. Select **Edit | Delete Map Model** and click **Yes** in the Attention dialog that opens. Close the ImageInfo dialog, and begin this exercise.

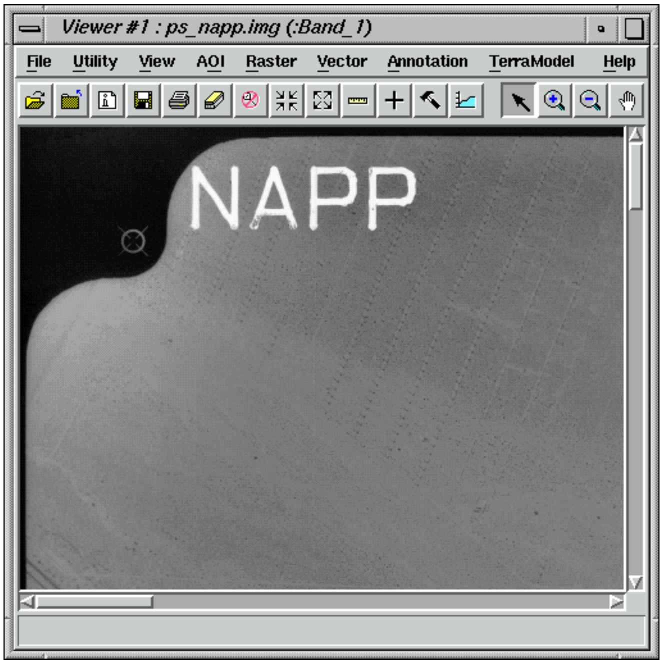
1. Select **File | Open | Raster Layer** from the Viewer menu bar, or click the Open icon in the Viewer tool bar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, select **ps_napp.img** from the directory into which you copied it.

3. Click **OK** in the Select Layer To Add dialog to display the camera image in the Viewer.

   The file **ps_napp.img** opens in the Viewer. The Viewer image is displayed with a view of the top, left corner of the photo, as shown in the following picture:

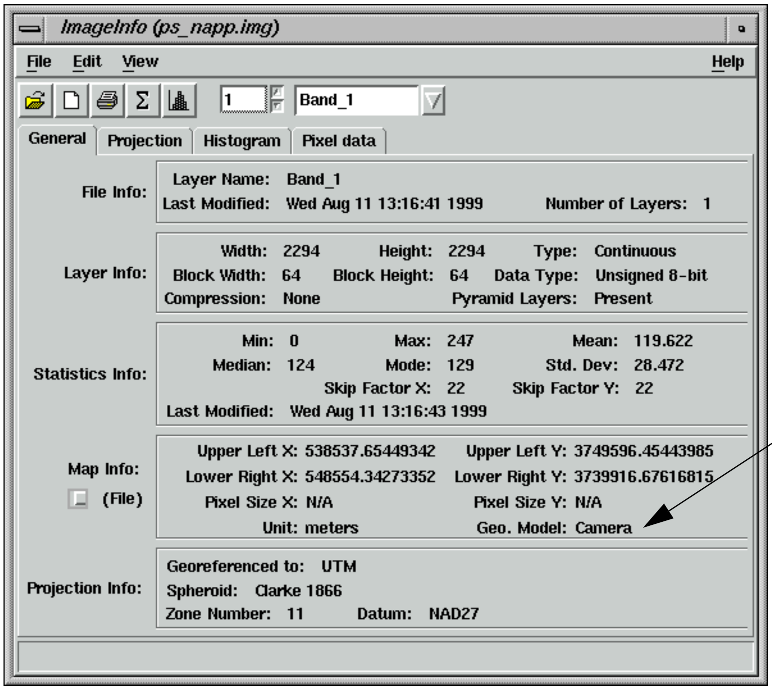


Check for Map Model

Before you continue with geometric correction, you must first make sure that the image does not already have a map model.

1. From the Viewer tool bar, click the ImageInfo icon.

The ImageInfo dialog opens.



File Info:
Layer Name: Band_1
Last Modified: Wed Aug 11 13:16:41 1999
Number of Layers: 1

Layer Info:
Width: 2294
Height: 2294
Type: Continuous
Block Width: 64
Block Height: 64
Data Type: Unsigned 8-bit
Compression: None
Pyramid Layers: Present

Statistics Info:
Min: 0
Max: 247
Mean: 119.622
Median: 124
Mode: 129
Std. Dev: 28.472
Skip Factor X: 22
Skip Factor Y: 22
Last Modified: Wed Aug 11 13:16:43 1999

Map Info: ☐ (File)
Upper Left X: 538537.65449342
Upper Left Y: 3749596.45443985
Lower Right X: 548554.34273352
Lower Right Y: 3739916.67616815
Pixel Size X: N/A
Pixel Size Y: N/A
Unit: meters
Geo. Model: Camera

Projection Info:
Georeferenced to: UTM
Spheroid: Clarke 1866
Zone Number: 11
Datum: NAD27

The model is listed here

2. Look in the **Geo. Model** section. If the **Geo. Model** section says **Camera**, you must delete the map model, therefore, proceed to step 3. If there is no model, proceed to "Perform Geometric Correction" on page 268.

3. Select **Edit | Delete Map Model** from the ImageInfo menu bar.

4. Select **File | Close** from the ImageInfo menu bar.

Redisplay the file

1. Click the Close icon in the Viewer currently displaying **ps_napp.img**.

2. Click the Open icon, and select **ps_napp.img** from the directory in which you saved it.

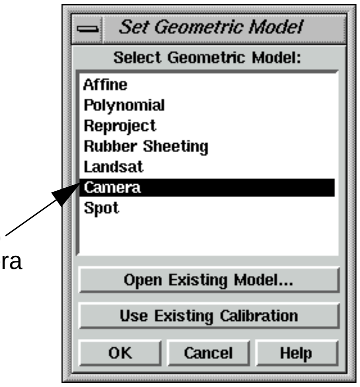
3. Move your cursor around the image and note the small coordinates in the status area.

   The small coordinates are pixel coordinates, not map coordinates. You can now proceed with geometric correction.

Perform Geometric Correction

1. Select **Raster | Geometric Correction** from the Viewer menu bar.

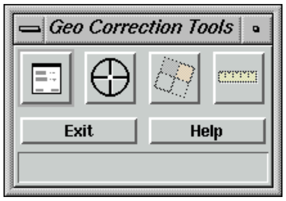
   The Set Geometric Model dialog opens.


Set Geometric Model
Select Geometric Model:
Affine
Polynomial
Reproject
Rubber Sheeting
Landsat
Camera
Spot
Open Existing Model...
Use Existing Calibration
OK
Cancel
Help

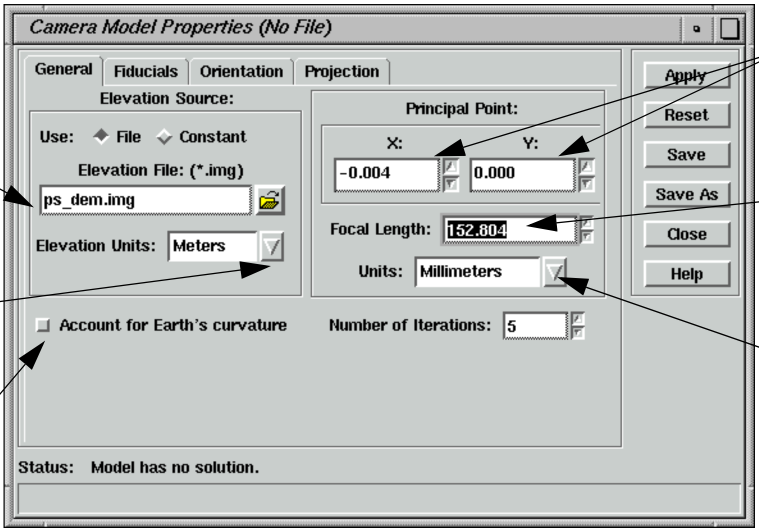
Click here to select Camera

2. In the Set Geometric Model dialog, click on **Camera**, and then click **OK**.

   The Geo Correction Tools open.


Geo Correction Tools
Exit
Help

The Camera Model Properties dialog also opens.



Elevation Source:

Use: ◆ File ◇ Constant

Elevation File: (*.img) ps_dem.img

Elevation Units: Meters

Principal Point:

X: -0.004

Y: 0.000

Focal Length: 152.804

Units: Millimeters

☐ Account for Earth's curvature

Number of Iterations: 5

Status: Model has no solution.

Enter the DEM file name here

Click here to select the elevation units

Check to be sure that this checkbox is deactivated

Enter the X and Y principal point coordinates here

Enter the focal length here

Select the units for the principal points and focal length here

Set Camera Model Properties

1. In the Camera Model Properties dialog, enter the DEM file (**ps_dem.img**) under **Elevation File**.

   NOTE: Upon request, the data provider supplies the camera calibration certificate with the film at the time of purchase. This certificate provides the information needed for steps (2. and 3.

2. In the Camera Model Properties dialog under **Principal Point**, enter **-0.004** for **X** and accept **0.000** as the default for **Y**. Then enter **152.804** for the **Focal Length**.

   NOTE: From the camera calibration certificate, there may be several possible types of the Principal Point coordinates. The Principal Point of Symmetry is preferable.

3. In **Units** under **Principal Point**, accept the default of **Millimeters**.

   NOTE: The **X** *and* **Y Principal Point** *coordinates,* **Focal Length**, *and* **Fiducial Film** *coordinates must all be entered in the same units.*

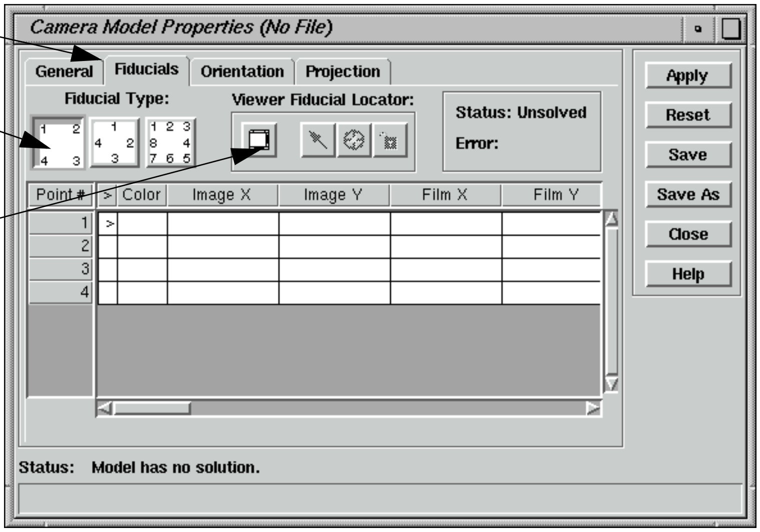
4. For this example, make sure that the **Account for Earth's curvature** checkbox not selected.

   You should only account for the Earth's curvature when using small-scale images or when it is necessary to take this factor into account. Alternately activating and deactivating this option (and then clicking **Apply**) *allows you to observe changes to the RMS error. Accounting for the Earth's curvature slows down the rectification process.*

Edit Fiducials

1. Click on the **Fiducials** tab at the top of the Camera Model Properties dialog.

   The Fiducial options display.


Camera Model Properties (No File)
General
Fiducials
Orientation
Projection
Fiducial Type:
Viewer Fiducial Locator:
Status: Unsolved
Error:
Point #
>
Color
Image X
Image Y
Film X
Film Y
1
2
3
4
Apply
Reset
Save
Save As
Close
Help

Status: Model has no solution.

Click here to view the Fiducials options

Click here to select this Fiducial Type

Click here to select the Viewer in which to locate fiducials

2. Under **Fiducial Type**, click the first icon.

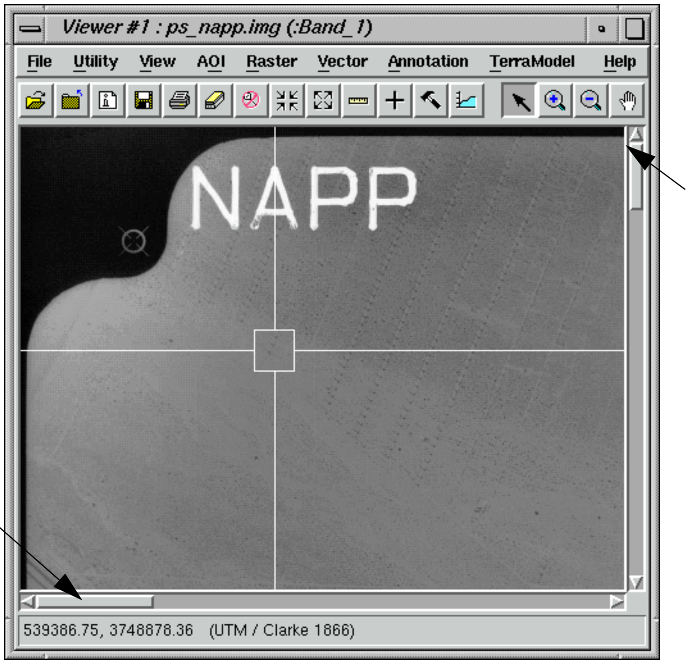
3. Under **Viewer Fiducial Locator**, click the Toggle icon.

4. Follow the instructions by clicking in the Viewer that contains **ps_napp.img**.

   A link box opens in the first Viewer, and the Chip Extraction Viewer also displays (the second Viewer).

5. In the first Viewer, drag the link box to the point in the image you want to digitize (as illustrated in the following example). Place the center of the link box on the dot at the center of the point (where the crosshair intersects).

   NOTE: Identifying the dot may sometimes require Breakpoint/LUT adjustments.


Viewer #1 : ps_napp.img (:Band_1)
File Utility View AOI Raster Vector Annotation TerraModel Help
NAPP
539386.75, 3748878.36 (UTM / Clarke 1866)

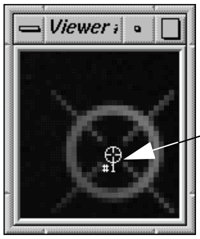
Drag on this scroll bar to view the top or bottom of the image

Drag on this scroll bar to view the left and right sides of the image

The second Viewer displays the point in the image that you have chosen with the link box.

6. In the Camera Model Properties dialog, click the Place Image Fiducial icon.

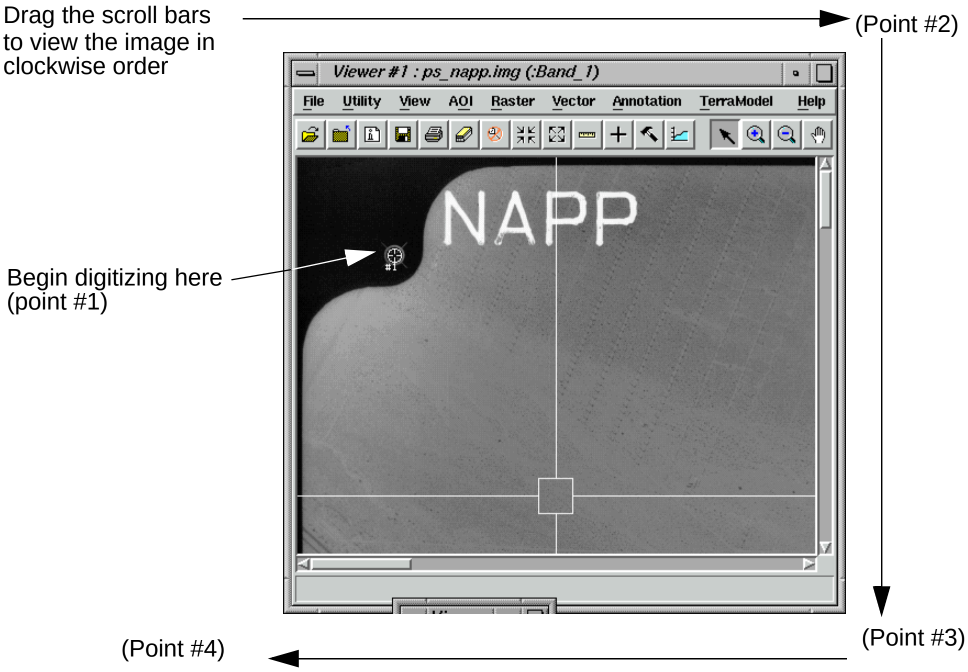
Move your cursor into the Chip Extraction Viewer (the second Viewer), and click on the center point where the crosshair intersects.



Click here to place the image fiducial

The point coordinates display under **Image X** and **Image Y** in the Fiducials CellArray of the Camera Model Properties dialog.

7. Create three more fiducials by repeating steps (5. and 6. for the points in the three other corners of the image in the first Viewer. Move clockwise around the image in the Viewer, using the Viewer scroll bars, as shown in the following diagram:


Viewer #1 : ps_napp.img (:Band_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
NAPP

Drag the scroll bars to view the image in clockwise order

(Point #2)

Begin digitizing here (point #1)

(Point #3)

(Point #4)

It is crucial that you follow step 7. exactly to match the points with the coordinates in step 1.

Enter Film Coordinates

1. It is necessary to enter the **Film** coordinates into the Camera Model Properties dialog manually. The data provider can include this information in the camera calibration certificate.

   Being sure to match the point numbers with the proper coordinates, enter the **Film X** and **Film Y** coordinates from the following table:

Table 12-1: Film X and Film Y Coordinates

Point #	Film X	Film Y
1	-106.000	106.000
2	105.999	105.994
3	105.998	-105.999
4	-106.008	-105.999

When the last **Film** coordinate has been entered in the Camera Model Properties dialog, the **Status** changes to **Solved** and the **Error** is calculated. This indicates that the interior orientation parameters have been solved.

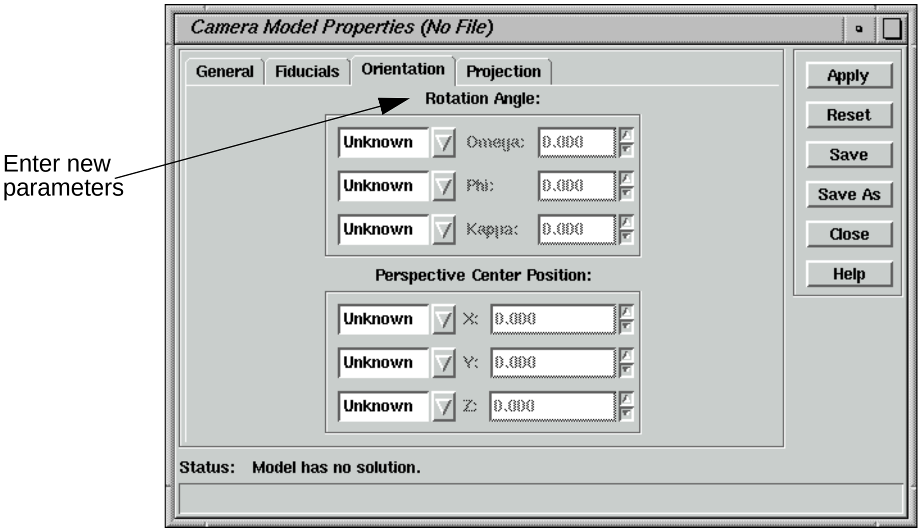
An error of less than **1.0000** is acceptable. An error of greater than **1.0000** indicates that the points were inaccurately measured or poorly identified.

2. In the Camera Model Properties dialog under **Viewer Fiducial Locator**, click the Toggle icon.

The Chip Extraction Viewer (the second Viewer) closes.

3. Click on the **Orientation** tab in the Camera Model Properties dialog.

The Orientation options display.



Camera Model Properties (No File)

Rotation Angle:

Unknown Omega: 0.000

Unknown Phi: 0.000

Unknown Kappa: 0.000

Perspective Center Position:

Unknown X: 0.000

Unknown Y: 0.000

Unknown Z: 0.000

Status: Model has no solution.

If you have known parameters for the **Rotation Angle** and **Perspective Center Position** derived from another triangulation package, or if you have estimated values, you can enter them in the **Orientation** tab. Select the appropriate settings: **Unknown**, **Estimate**, or **Fixed**.

If no changes are being made to the **Orientation**, you do not need to click **Apply** (therefore, it is deactivated).

NOTE: If **Account for Earth's curvature** *is selected under the* **General** *tab, then the Orientation options are disabled (see step 4. on page 269).*

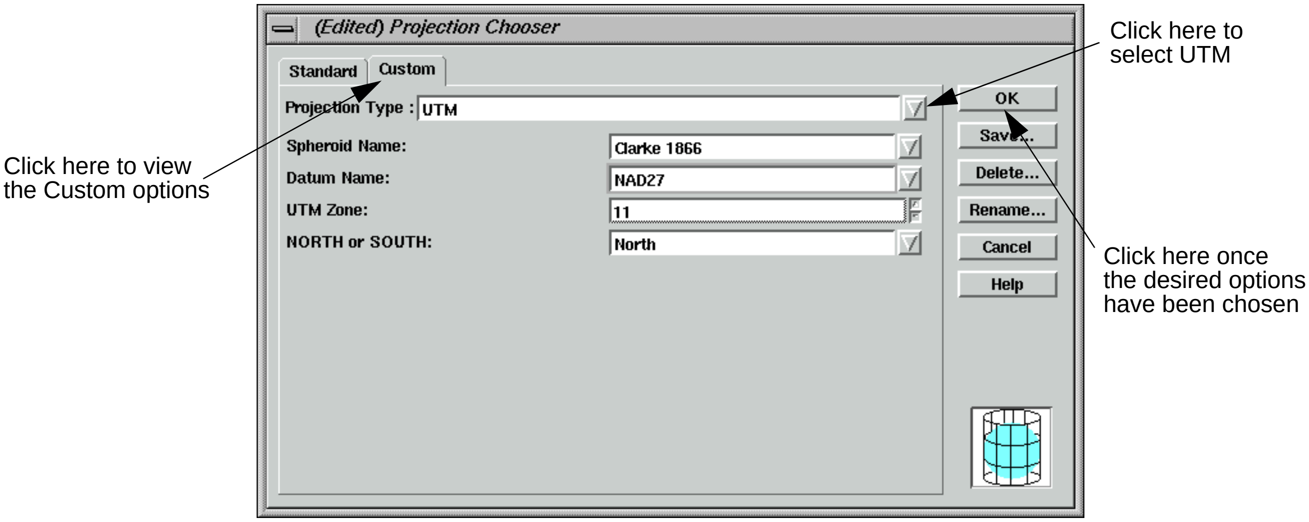
Change Projection

1. Click on the **Projection** tab in the Camera Model Properties dialog.

The Projection options display.

2. In the Projection options, click on **Add/Change Projection**.

The Projection Chooser dialog opens.



Projection Type : UTM

Spheroid Name: Clarke 1866

Datum Name: NAD27

UTM Zone: 11

NORTH or SOUTH: North

Click here to view the Custom options

Click here to select UTM

Click here once the desired options have been chosen

3. In the Projection Chooser dialog, click on the **Custom** tab.

4. Confirm that **Projection Type** lists **UTM**.

5. Confirm that **Datum Name** lists **NAD27**.

6. Confirm that **11** is the **UTM Zone**.

7. Click **OK** in the Projection Chooser dialog.

The projection information you just entered displays under **Current Reference Map Projection** in the Camera Model Properties dialog.

8. In the Camera Model Properties dialog, click on the popup list next to **Map Units** and select **Meters** (this activates the **Apply** button).

9. Click **Apply** and then **Save As** in the Camera Model Properties dialog.

Name the Geometric Model

1. The Geometric Model Name dialog opens.

2. In the Geometric Model Name dialog under **Filename**, enter the name **GeoModel** in the directory of your choice. The .gms file extension is added automatically.

3. Click **OK** in the Geometric Model Name dialog.

Start the GCP Tool and compute RMS Error

1. In the Geo Correction Tools, click on the GCP Tool icon.

   The GCP Tool Reference Setup dialog opens.

2. In the GCP Tool Reference Setup dialog, select **GCP File** under **Collect Reference Points From**, and then click **OK**.

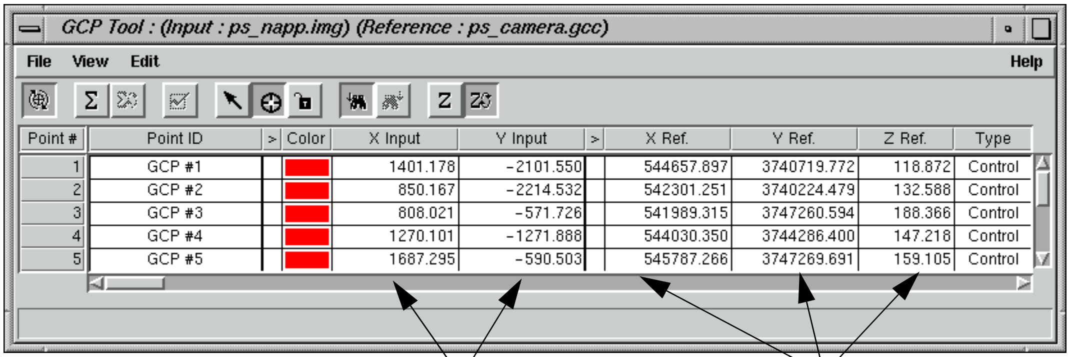
   The Reference GCC File dialog opens.

3. In the Reference GCC File dialog under **Filename**, select **ps_camera.gcc**.

   The reference points in this file were obtained from USGS 1:24,000 scale topographical maps using a digitizer.

4. Click **OK** in the Reference GCC File dialog.

   A Chip Extraction Viewer (the second Viewer), a link box, and the GCP Tool open.


GCP Tool : (Input : ps_napp.img) (Reference : ps_camera.gcc)
File
View
Edit
Help

Point #	Point ID	>	Color	X Input	Y Input	>	X Ref.	Y Ref.	Z Ref.	Type
1	GCP #1		🟥	1401.178	-2101.550		544657.897	3740719.772	118.872	Control
2	GCP #2		🟥	850.167	-2214.532		542301.251	3740224.479	132.588	Control
3	GCP #3		🟥	808.021	-571.726		541989.315	3747260.594	188.366	Control
4	GCP #4		🟥	1270.101	-1271.888		544030.350	3744286.400	147.218	Control
5	GCP #5		🟥	1687.295	-590.503		545787.266	3747269.691	159.105	Control

These file coordinates were previously measured and saved in the source file, ps_napp.img

These reference coordinates are from the ps_camera.gcc file

5. In the GCP Tool, click on the Calculate icon.

   NOTE: The orthorectification models do not have the option of Automatic Transform Calculation.

Clicking this icon solves the model and calculates the RMS error and residuals. The **Control Point Error** for the **X** and **Y** values is displayed in the upper, right-hand corner of the GCP Tool.

NOTE: The GCP Tool requires a minimum of three GCPs to run the model and at least six GCPs to make the model accurate and stable.

6. Click **Save** in the Camera Model Properties dialog.

Resampling vs. Calibration

Resampling

Resampling is the process of calculating the file values for the rectified image and creating the new file. All of the raster data layers in the source file are resampled. The output image has as many layers as the input image.

ERDAS IMAGINE provides these widely-known resampling algorithms:

- Nearest Neighbor
- Bilinear Interpolation
- Cubic Convolution
- Bicubic Spline

Calibration

Instead of creating a new, rectified image by resampling the original image based on the mathematical model, calibrating an image only saves the mathematical model into the original image as a piece of auxiliary information. Calibration does not generate new images, so when the calibrated image is used, the math model comes into play as needed.

For example, if you want to see the calibrated image in its rectified map space in a Viewer, the image can be resampled on the fly based on the math model, by selecting the **Orient image to map system** option in the Select Layer To Add dialog.

A major drawback to image calibration is that the processes involved with the calibrated image is slowed down significantly if the math model is complicated. One minor advantage to image calibration is that it uses less disk space and leaves the image's spectral information undisturbed.

It is recommended that image calibration be used only when necessary, due to the drawbacks of the process.

Choose Your Path

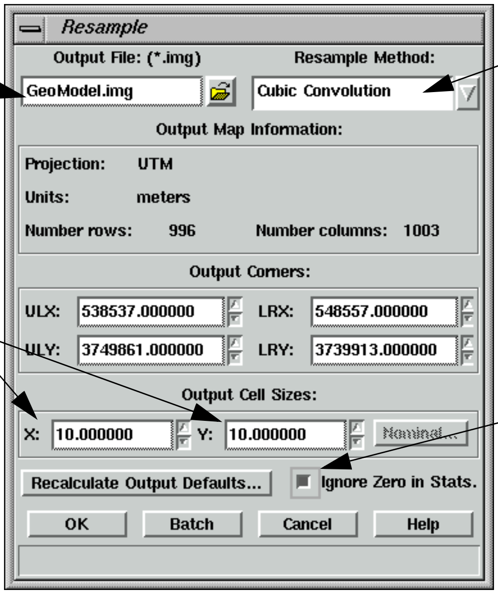
- If you would like to resample the camera image, proceed to "Resample the Image" on page 277.
- If you would like to calibrate the camera image, proceed to "Calibrate the Image" on page 278.

Resample the Image

Resampling requires an input file and a transformation matrix by which to create the new pixel grid.

1. In the Geo Correction Tools, click on the Resample icon.

The Resample dialog opens.



Output File: (*.img) GeoModel.img

Resample Method: Cubic Convolution

Output Map Information:

Projection: UTM

Units: meters

Number rows: 996

Number columns: 1003

Output Corners:

ULX: 538537.000000

LRX: 548557.000000

ULY: 3749861.000000

LRY: 3739913.000000

Output Cell Sizes:

X: 10.000000

Y: 10.000000

☐ Ignore Zero in Stats.

Enter the output file name here

Click on this popup list to select Cubic Convolution

Change the output cell sizes here

Click to enable this option

2. In the Resample dialog under **Output File**, enter **GeoModel.img** in the directory of your choice.

3. Under **Resample Method**, click on the popup list and select **Cubic Convolution**.

4. Under **Output Cell Sizes**, enter **10** for **X** and **10** for **Y**.

   NOTE: The default **Output Cell Sizes** *are based on the triangulation. The smaller the pixel size, the larger the output file size.*

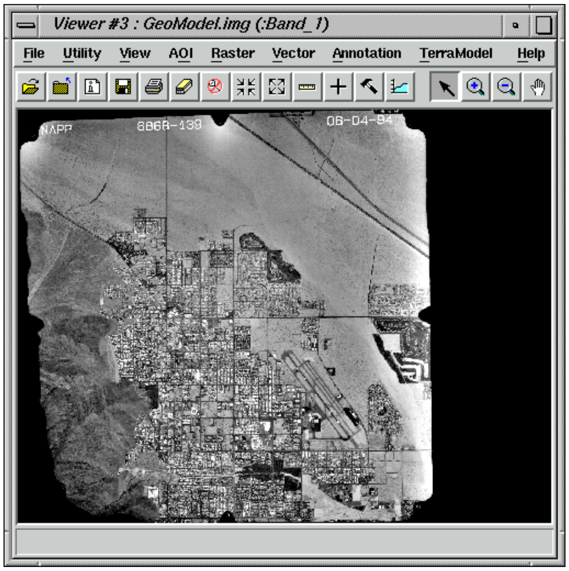
5. Click the **Ignore Zero in Stats.** checkbox to activate it.

6. Click **OK** in the Resample dialog.

   A Job Status dialog displays, indicating the progress of the function.

7. When the Job Status dialog indicates that the process is 100% complete, click **OK**.

8. Display **GeoModel.img** in a Viewer to view the resampled orthoimage.



Calibrate the Image

You must have completed the steps to rectify **ps_napp.img** on pages 265 through 276.

1. In the Geo Correction Tools, click on the Calibrate Image icon.

   A Calibrate Image warning box displays, indicating that “performing image calibration requires the termination of the Geo Correction Tool and reopening the image in the Viewer.”

2. Click **OK** in the Calibrate Image warning box.

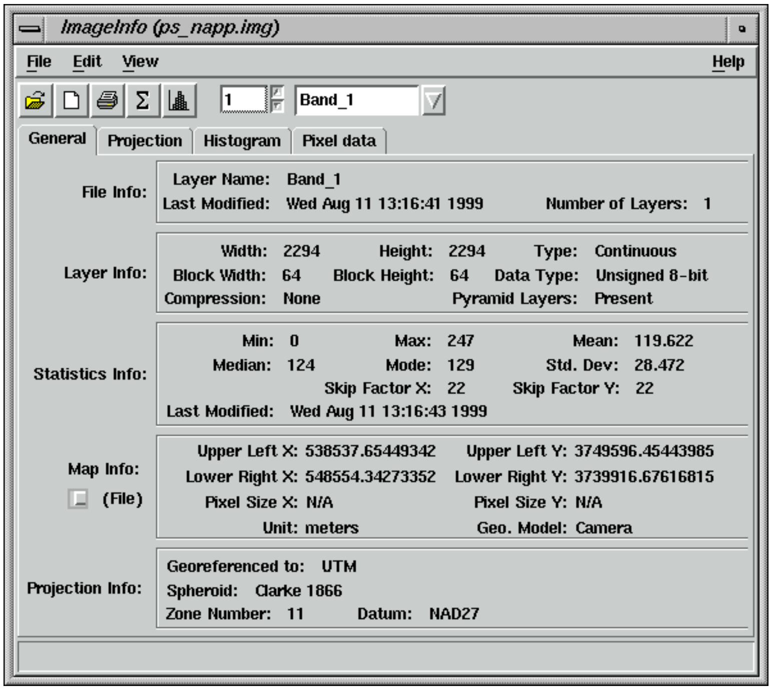
   The Geo Correction Tool and all its associated dialogs close. The file **ps_napp.img** closes and then reopens in the Viewer, with the **Orient Image to Map System** option turned off.

3. To apply the calibration to the image in the Viewer, redisplay the output image with the **Orient Image to Map System** option turned on.

   *NOTE: Once calibrated, this image cannot be reused in the orthorectification process using the information/coordinates files provided. Calibration must be deleted (***Edit | Delete Map Model*** in the ImageInfo dialog) for this file to be used again for this tour guide.*

4. In the Viewer, click on the Info icon to view the calibration information.

   The ImageInfo dialog opens, displaying the information for the calibrated image.


ImageInfo (ps_napp.img)
File
Edit
View
Help
1
Band_1
General
Projection
Histogram
Pixel data

File Info:
Layer Name: Band_1
Last Modified: Wed Aug 11 13:16:41 1999
Number of Layers: 1

Layer Info:
Width: 2294
Height: 2294
Type: Continuous
Block Width: 64
Block Height: 64
Data Type: Unsigned 8-bit
Compression: None
Pyramid Layers: Present

Statistics Info:
Min: 0
Max: 247
Mean: 119.622
Median: 124
Mode: 129
Std. Dev: 28.472
Skip Factor X: 22
Skip Factor Y: 22
Last Modified: Wed Aug 11 13:16:43 1999

Map Info: ☐ (File)
Upper Left X: 538537.65449342
Upper Left Y: 3749596.45443985
Lower Right X: 548554.34273352
Lower Right Y: 3739916.67616815
Pixel Size X: N/A
Pixel Size Y: N/A
Unit: meters
Geo. Model: Camera

Projection Info:
Georeferenced to: UTM
Spheroid: Clarke 1866
Zone Number: 11
Datum: NAD27

For a more in-depth discussion of the concepts behind rectification, see "Chapter 9: Rectification" *of the* ERDAS Field Guide.

CHAPTER THIRTEEN

Terrain Surface Interpolation

Introduction

The Surfacing Tool enables you to create a three-dimensional surface from irregularly spaced points. Supported input data include:

- ASCII point files
- Arc point and line coverages
- ERDAS IMAGINE *.ovr layers
- existing raster images (IMG)

All input data sources must have X, Y, and Z values. Surface Interpolation calculates Z values at spatial locations where no Z samples exist in the input data source. The output is a continuous raster image that contains Z values calculated from the interpolation process.

The ERDAS IMAGINE Surface Tool uses a TIN interpolation method. At each point where there is a known value, that known value remains unchanged in the output surface. Where the value is not known, it is interpolated from the surrounding known values.

Two TIN interpolation methods are available in the Surface Tool: Linear and Nonlinear. The Linear interpolation method, which makes use of a first-order polynomial equation, results in the TIN triangles being defined as angular planes. The Nonlinear interpolation method, which uses a fifth-order polynomial, results in a smooth surface. In this case, the TIN triangle areas are not considered to be planes, but areas that have rubber sheet characteristics. The Linear interpolation method is quicker and the results more predictable. However, the Nonlinear interpolation method produces more continuous results from irregularly distributed data sets where the observed phenomena has a rolling, nonangular surface characteristic.

Approximate completion time for this tour guide is 15 minutes.

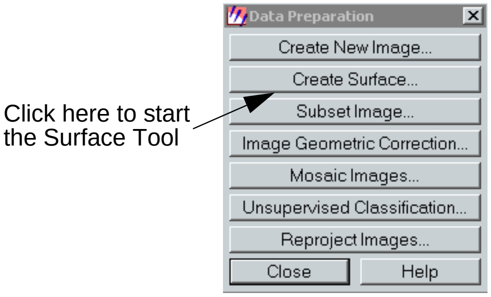
Create a Surface

ERDAS IMAGINE must be running and a Viewer open.

1. Click on the DataPrep icon on the ERDAS IMAGINE icon panel.

DataPrep

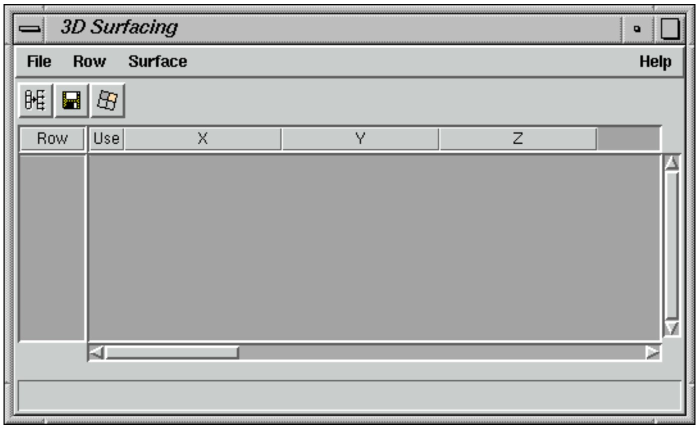
The **Data Preparation** menu opens.


Data Preparation
Create New Image...
Create Surface...
Subset Image...
Image Geometric Correction...
Mosaic Images...
Unsupervised Classification...
Reproject Images...
Close
Help

Click here to start the Surface Tool

2. In the **Data Preparation** menu, click **Create Surface**.

   The 3D Surfacing dialog opens.

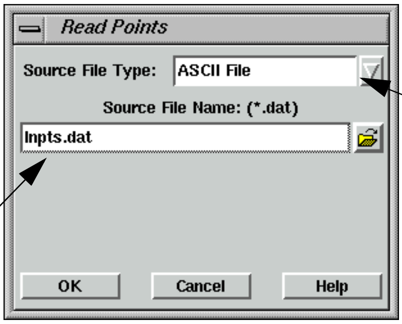

3D Surfacing
File
Row
Surface
Help
Row
Use
X
Y
Z

3. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Import an ASCII File

1. In the 3D Surfacing dialog, click the Read Points icon.

   The Read Points dialog opens.


Read Points
OK
Cancel
Help

Source File Type: ASCII File

Source File Name: (*.dat)

Inpts.dat

Click this popup and select ASCII File

Enter the name of the source file here

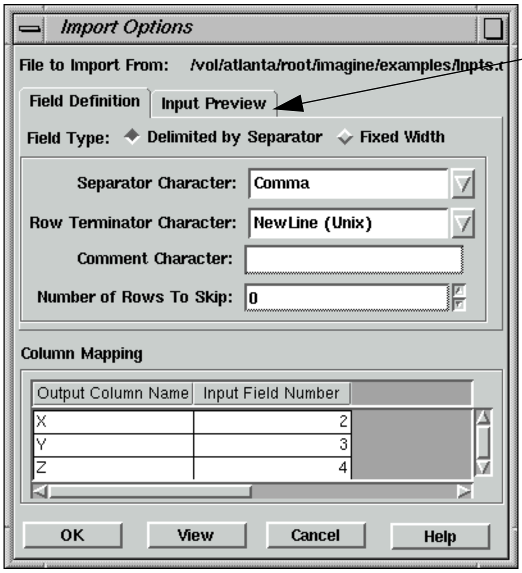
2. In the Read Points dialog, click on the popup next to **Source File Type** and select **ASCII File**.

3. Under **Source File Name**, enter **Inpts.dat**.

This file is located in the <IMAGINE_HOME>/examples directory, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

4. Click **OK** in the Read Points dialog.

The Read Points dialog closes, and the Import Options dialog opens.


Import Options
Field Definition
Input Preview
Column Mapping
OK
View
Cancel
Help

File to Import From: /vol/atlanta/root/imagine/examples/lnpts.

Field Type: ● Delimited by Separator ○ Fixed Width

Separator Character: Comma

Row Terminator Character: NewLine (Unix)

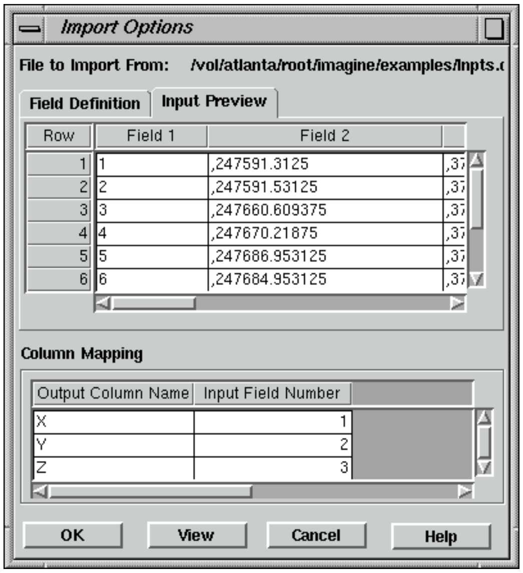
Comment Character:

Number of Rows To Skip: 0

Output Column Name	Input Field Number
X	2
Y	3
Z	4

Click on this tab to preview the input data

5. In the Import Options dialog, click the **Input Preview** tab to see how the ASCII file is imported and mapped under the present parameter settings.



File to Import From: /vol/atlanta/root/imagine/examples/lnpts.

Row	Field 1	Field 2	
1	1	,247591.3125	,37
2	2	,247591.53125	,37
3	3	,247660.609375	,37
4	4	,247670.21875	,37
5	5	,247686.953125	,37
6	6	,247684.953125	,37

Output Column Name	Input Field Number
X	1
Y	2
Z	3

From the **Input Preview** display, you can tell that the **Separator Character** is the comma and that **Field 1** should be ignored.

6. Click the **Field Definition** tab.

7. Click on the popup list next to **Separator Character** and select **Comma**.

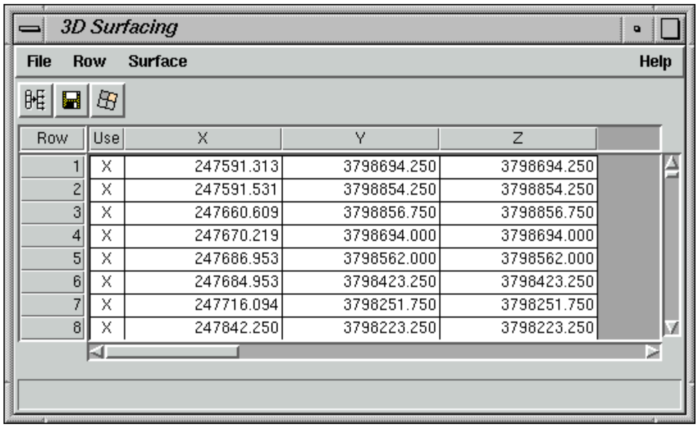
8. In the **Column Mapping** CellArray, alter the **Input Field Number** column values vertically from **1, 2, 3** to **2, 3, 4** in order to ignore the ID column of the input file.

9. Click **OK** in the Import Options dialog.

   A Job Status dialog opens, reporting the progress of the function.

10. When the Job Status dialog shows that the process is 100% complete, click **OK**.

The **X**, **Y**, and **Z** columns of the 3D Surfacing CellArray are now populated with 4,411 rows of X, Y, and Z coordinates.



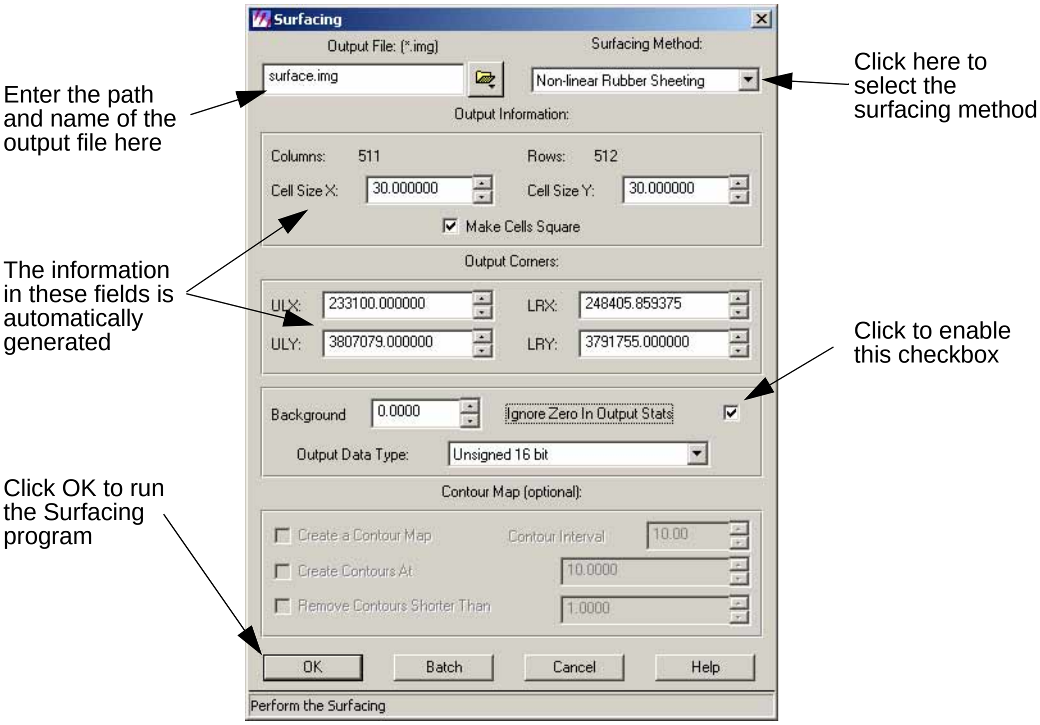
Row	Use	X	Y	Z
1	X	247591.313	3798694.250	3798694.250
2	X	247591.531	3798854.250	3798854.250
3	X	247660.609	3798856.750	3798856.750
4	X	247670.219	3798694.000	3798694.000
5	X	247686.953	3798562.000	3798562.000
6	X	247684.953	3798423.250	3798423.250
7	X	247716.094	3798251.750	3798251.750
8	X	247842.250	3798223.250	3798223.250

11. If you like, you can save these points as a **Point Coverage** (.arcinfo) or an **Annotation Layer** (.ovr) by selecting **File | Save As** from the 3D Surfacing dialog menu bar.

Perform Surfacing

1. In the 3D Surfacing dialog, click the Perform Surfacing icon.

The Surfacing dialog opens. The extent and cell size defaults are filled in automatically, based on the source ASCII file.



Surfacing

Output File: (*.img) surface.img

Surfacing Method: Non-linear Rubber Sheeting

Output Information:

Columns: 511

Rows: 512

Cell Size X: 30.000000

Cell Size Y: 30.000000

☑ Make Cells Square

Output Corners:

ULX: 233100.000000

LRX: 248405.859375

ULY: 3807079.000000

LRY: 3791755.000000

Background 0.0000

Ignore Zero In Output Stats ☑

Output Data Type: Unsigned 16 bit

Contour Map (optional):

☐ Create a Contour Map

Contour Interval 10.00

☐ Create Contours At 10.0000

☐ Remove Contours Shorter Than 1.0000

Perform the Surfacing

Enter the path and name of the output file here

Click here to select the surfacing method

The information in these fields is automatically generated

Click to enable this checkbox

Click OK to run the Surfacing program

The two options for a surfacing method are **Linear Rubber Sheeting** (1st Order Polynomial solution) and **Non-linear Rubber Sheeting** (5th Order Polynomial solution).

2. Under **Output File**, enter the name of the output file (e.g., **surface.img**) in the directory of your choice.

3. Click on the popup list next to **Surfacing Method** and select **Non-linear Rubber Sheeting**.

4. Click the **Ignore Zero In Output Stats** checkbox to enable it.

5. Click **OK** in the Surfacing dialog.

   A Job Status dialog displays, stating the progress of the function.

6. When the Job Status dialog reads that the function is 100% complete, click **OK** (if necessary).

Display the Surface

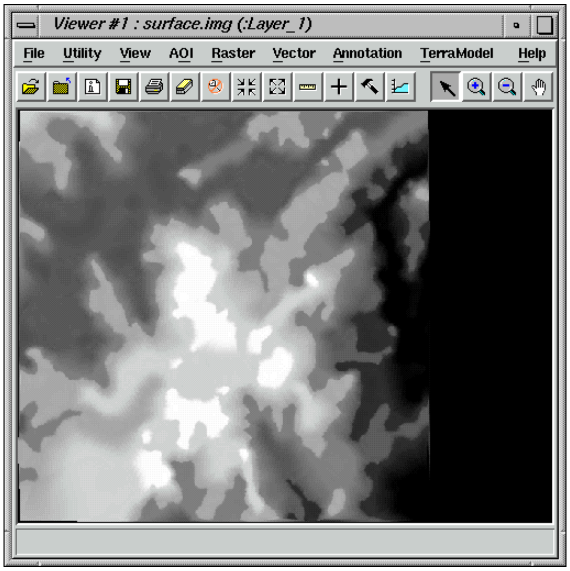
1. Click the Open icon in a Viewer.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, enter the name of the output file you created in step 2., beginning with the directory path in which you saved it.

3. Click **OK** in the Select Layer To Add dialog.

   The output image is displayed in the Viewer for you to examine.


Viewer #1 : surface.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

To edit portions of the resulting surface, use the raster editing techniques described in "Chapter 1: Viewer", "Raster Editor" *on page 33.*

CHAPTER FOURTEEN

Mosaic

Introduction

This tour guide gives you the steps for mosaicking two or more image files to produce one image file. The mosaicking process works with rectified and/or calibrated images. Here, you are shown how to mosaic air photo images as well as LANDSAT images.

Approximate completion time for this tour guide is 45 minutes.

Mosaic Using Air Photo Images

In this section, you use a template to mosaic two air photo images.

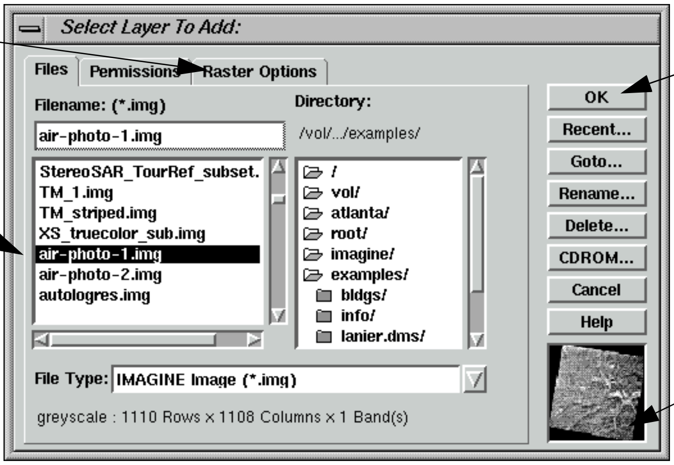
The two files to be mosaicked are **air-photo-1.img** and **air-photo-2.img**.

These data files are air photo images of the Oxford, Ohio area.

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer, select **File | Open | Raster Layer** or click the Open icon.

The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
StereoSAR_TourRef_subset.
TM_1.img
TM_striped.img
XS_truecolor_sub.img
air-photo-1.img
air-photo-2.img
autologres.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/

Filename: (*.img) air-photo-1.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

greyscale : 1110 Rows × 1108 Columns × 1 Band(s)

Click here to access the raster options

Click here to select the file

Click here to display the file

Preview window

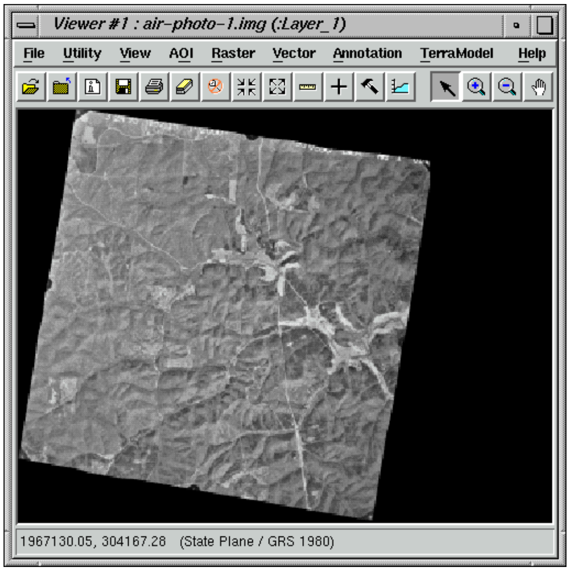
2. In the Select Layer To Add dialog under **Filename**, select **air-photo-1.img** from the file list.

3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

4. Click the **Fit to Frame** option to enable it.

5. Click **OK** in the Select Layer To Add dialog.

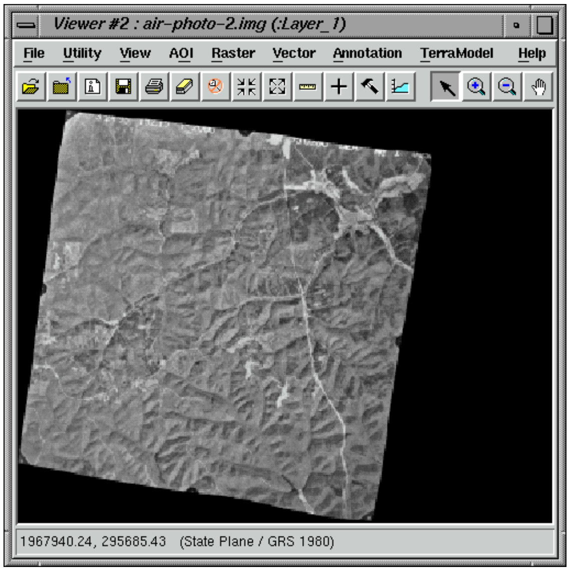
The file **air-photo-1.img** displays in the Viewer.


Viewer #1 : air-photo-1.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

1967130.05, 304167.28 (State Plane / GRS 1980)

6. Click on the Viewer icon in the ERDAS IMAGINE icon panel to create a second Viewer.

7. Repeat steps 1. through 5. for the second Viewer, selecting **air-photo-2.img** this time.


Viewer #2 : air-photo-2.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

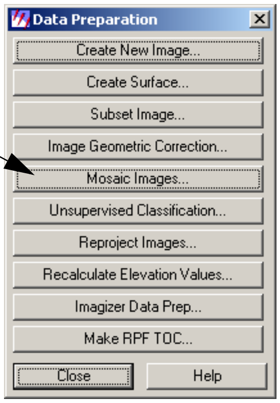
1967940.24, 295685.43 (State Plane / GRS 1980)

8. Position the Viewers side by side by selecting **Session | Tile Viewers** from the ERDAS IMAGINE menu bar.

   You can resize each Viewer so that they take up less screen space by dragging any corner of the Viewer.

9. In the ERDAS IMAGINE icon panel, click on the DataPrep icon.

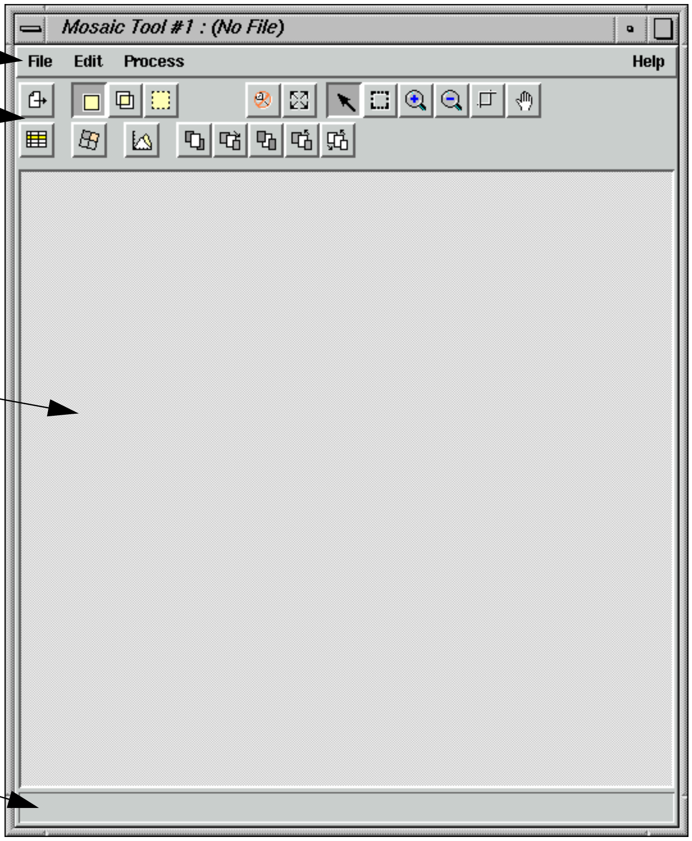
   The **Data Preparation** menu displays.


Data Preparation
Create New Image...
Create Surface...
Subset Image...
Image Geometric Correction...
Mosaic Images...
Unsupervised Classification...
Reproject Images...
Recalculate Elevation Values...
Imagizer Data Prep...
Make RPF TOC...
Close
Help

Click to start the Mosaic Tool

10. In the **Data Preparation** menu, select **Mosaic Images**.

    The Mosaic Tool viewer opens.


Mosaic Tool #1 : (No File)
File
Edit
Process
Help

Menu bar

Tool bar

Files to be mosaicked are displayed on the canvas

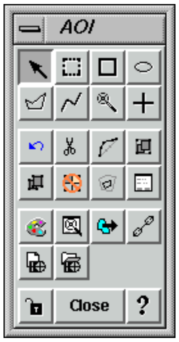
Status bar displays single-line help

11. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Set Input Images

1. In the first Viewer containing **air-photo-1.img**, select **AOI | Tools**.

The AOI tool palette displays.

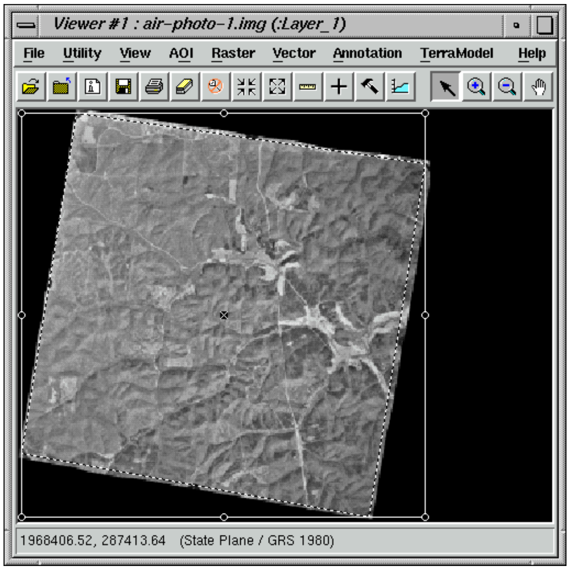


2. In the AOI tool palette, click on the Polygon icon.

3. In the first Viewer, draw a polygon around the entire image, outlining its shape, by dragging to draw lines and clicking to draw vertices (i.e., on-screen digitize the outline of the image). Middle-click to close the polygon.

NOTE: If you do not have a three-button mouse, you can double-click to close the polygon.

When you are finished, the AOI layer is highlighted with a dotted line and the image is surrounded by a bounding box. The image in the first Viewer should look like the following example:

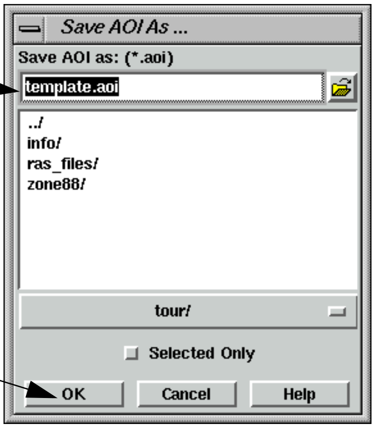

Viewer #1 : air-photo-1.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

1968406.52, 287413.64 (State Plane / GRS 1980)

Create a Template

1. When you have finished drawing the polygon, select **File | Save | AOI Layer As** in the first Viewer.

   The Save AOI As dialog opens.

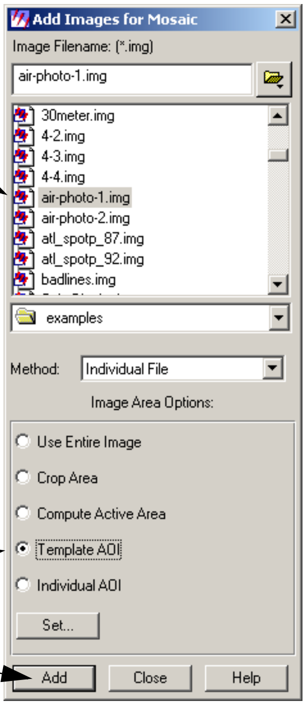

Save AOI As ...
Save AOI as: (*.aoi)
template.aoi
../
info/
ras_files/
zone88/
tour/
Selected Only
OK
Cancel
Help

Enter the directory and file name here

Click here to save the AOI

2. Under **Save AOI as**, enter **template.aoi** in the directory of your choice and click **OK**.
3. In the Mosaic Tool viewer, select **Edit | Add Images**.

   The Add Images for Mosaic dialog opens.


Add Images for Mosaic
Image Filename: (*.img)
air-photo-1.img
30meter.img
4-2.img
4-3.img
4-4.img
air-photo-1.img
air-photo-2.img
atl_spotp_87.img
atl_spotp_92.img
badlines.img
examples
Method:
Individual File
Image Area Options:
Use Entire Image
Crop Area
Compute Active Area
Template AOI
Individual AOI
Set...
Add
Close
Help

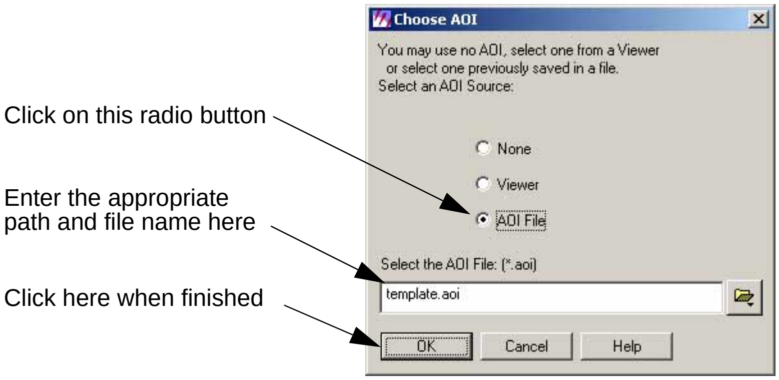
Select this file

Click this radio button and then click the accompanying Set button

4. Under **Image Filename**, select **air-photo-1.img**.

5. Under **Image Area Options**, click the **Template AOI** button, and then click the accompanying **Set** button.

   The Choose AOI dialog opens.



Click on this radio button

Enter the appropriate path and file name here

Click here when finished

Choose AOI

You may use no AOI, select one from a Viewer or select one previously saved in a file.
Select an AOI Source:

○ None
○ Viewer
● AOI File

Select the AOI File: (*.aoi)

template.aoi

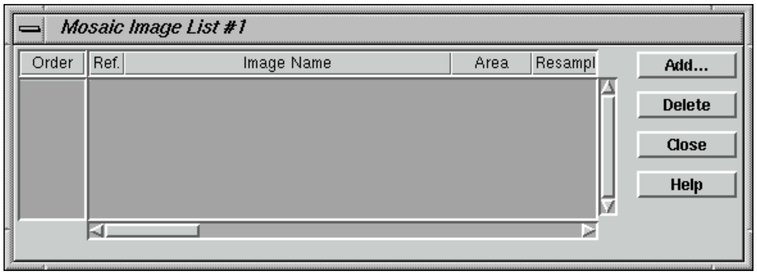
6. In the Choose AOI dialog under **AOI Source**, select **File**.

7. Under **AOI File**, select **template.aoi** from the directory in which you saved it.

8. Click **OK** in the Choose AOI dialog.

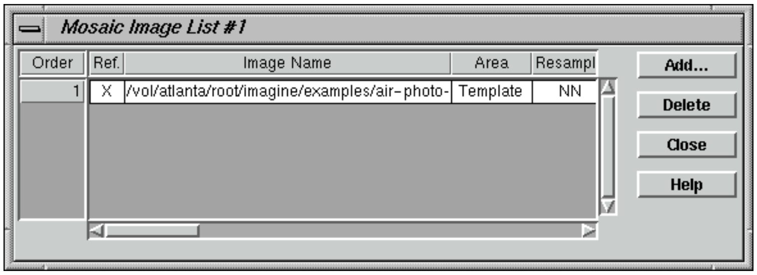
9. Select **Edit | Image List** from the Mosaic Tool viewer.

   The Mosaic Image List displays.



10. Click **Add** in the Add Images for Mosaic dialog.

    The data for **air-photo-1.img** display in the Mosaic Image List CellArray and a graphic of the image displays on the canvas of the Mosaic Tool viewer.


Mosaic Image List #1
Order
Ref.
Image Name
Area
Resampl
Add...
Delete
Close
Help

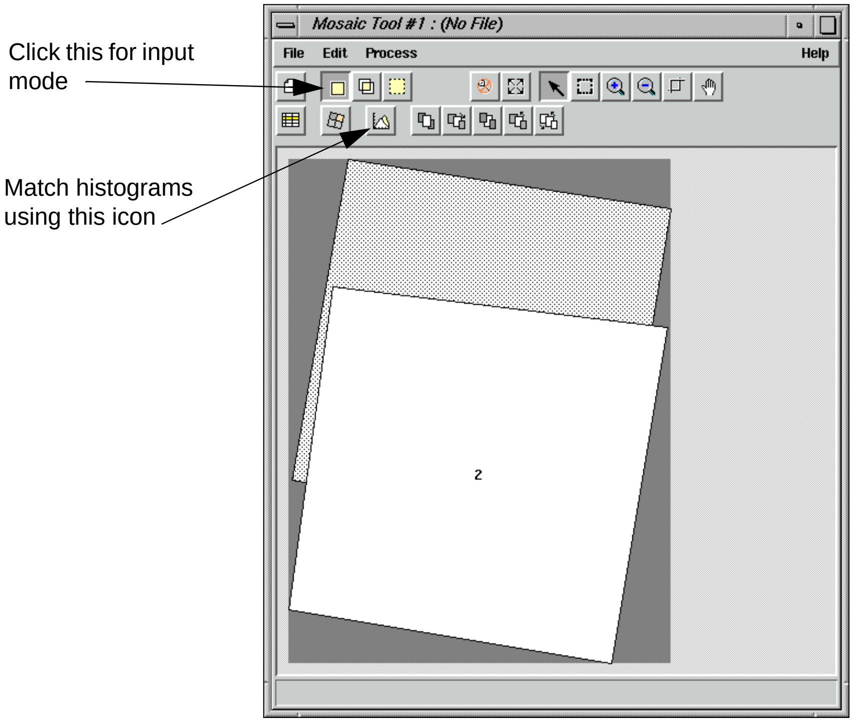
Order	Ref.	Image Name	Area	Resampl
1	X	/vol/atlanta/root/imagine/examples/air-photo-	Template	NN

11. In the Add Images for Mosaic dialog under **Image Filename**, select **air-photo-2.img**.

12. Under **Image Area Options**, click the **Compute Active Area** button.

13. Click **Add** and then **Close** in the Add Images for Mosaic dialog.

    The data for **air-photo-2.img** display in the Mosaic Image List CellArray, and a graphic of the image displays on top of the **air-photo-2.img** graphic in the canvas of the Mosaic Tool viewer.

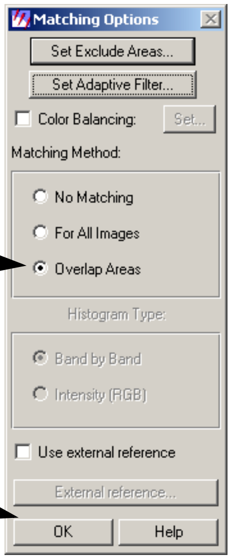

Click this for input mode
Match histograms using this icon
Mosaic Tool #1 : (No File)
File
Edit
Process
Help
2

Identify Areas of Intersection

1. In the Mosaic Tool viewer, click on the Input icon to verify that the input mode is activated.

2. Click on the Image Matching icon.

The Matching Options dialog opens.



Matching Options

☐ Color Balancing:

Matching Method:

○ No Matching
○ For All Images
● Overlap Areas

Histogram Type:

● Band by Band
○ Intensity (RGB)

☐ Use external reference

Click this radio button

Click here to apply the change

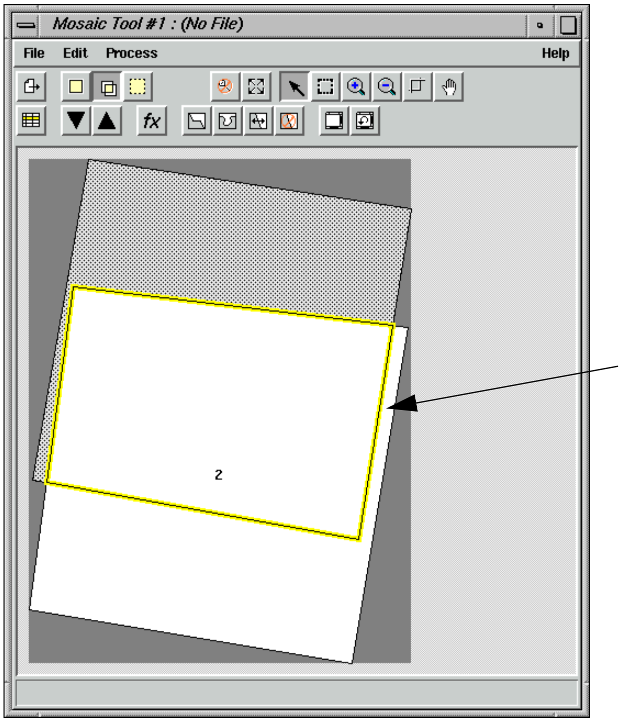
3. In the Matching Options dialog under **Matching Method**, click the **Overlap Areas** button.

4. Click **OK** in the Matching Options dialog.

5. In the Mosaic Tool viewer, click on the Intersection icon to activate the intersect mode.

6. In the canvas of the Mosaic Tool viewer, click on a line that intersects the two images.

   The lines that overlap the two images are highlighted in yellow.


Mosaic Tool #1 : (No File)
File
Edit
Process
Help
2

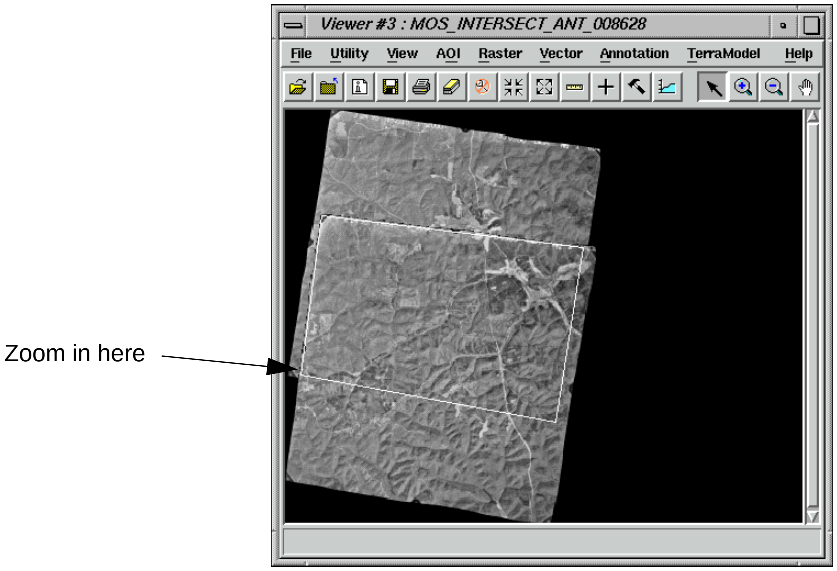
Click this line

Draw a Cutline

The Mosaic Tool enables you to draw one cutline through all the images or a single cutline in an individual image.

1. In the Mosaic Tool viewer, click on the Cutline Selection icon.

Viewer #3 opens, displaying the two images as they intersect.


Viewer #3 : MOS_INTERSECT_ANT_008628
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
Zoom in here

2. Use the Zoom In tool in the Viewer #3 tool bar to zoom in on the spot where you draw your cutline. (Do this by selecting the Zoom In tool and then clicking on the spot for which you want a closer view.)

3. Select the Line tool from the AOI tool palette.

   Once in the Viewer, the cursor becomes a crosshair.

4. In Viewer #3, draw the cutline by dragging inside the intersecting portion of the two images (the portion marked by the AOI box) (i.e., on-screen digitize the intersecting portion). Middle-click when finished.

   NOTE: When drawing cutlines, it is best to trace linear landmarks, such as rivers or roads. This conceals any seams in the resulting mosaic.

   NOTE: If you do not have a three-button mouse, you can double-click to close the polygon.

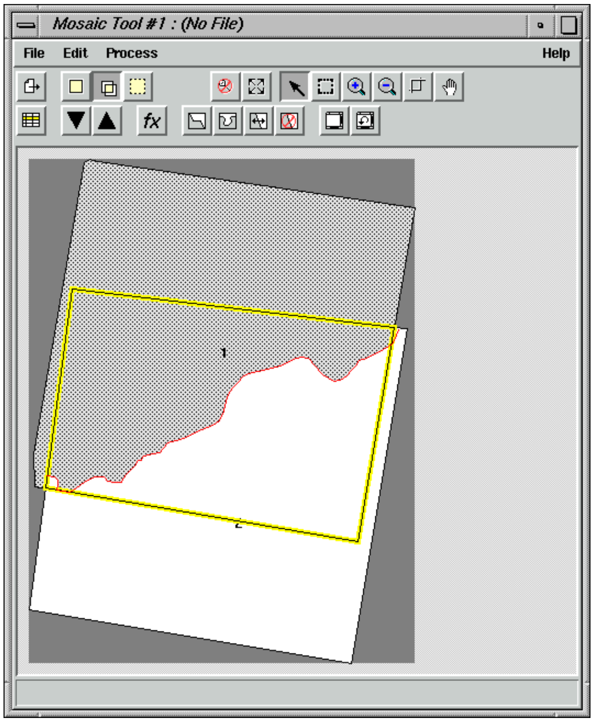
5. In the Mosaic Tool viewer, click on the AOI Cutline icon.

   The Choose AOI dialog opens.

6. In the Choose AOI dialog under **AOI Source**, select the **Viewer** button and click **OK**.

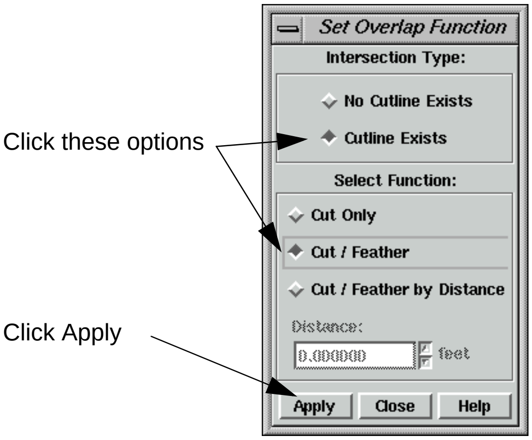
7. An Attention dialog opens, warning you that cutlines can be lost if projection is changed. Click **Yes** in the dialog.

   The cutline is highlighted in red in the Mosaic Tool viewer canvas.


Mosaic Tool #1 : (No File)
File
Edit
Process
Help
fx
1
2

8. Click the Function icon in the Mosaic Tool viewer.

   The Set Overlap Function dialog opens.


Set Overlap Function
Intersection Type:
○ No Cutline Exists
● Cutline Exists
Select Function:
○ Cut Only
● Cut / Feather
○ Cut / Feather by Distance
Distance:
0.000000
feet
Apply
Close
Help

Click these options

Click Apply

9. In the Set Overlap Function dialog under **Intersection Type**, select **Cutline Exists**.

10. Under **Select Function**, select **Cut/Feather**.

11. In the Set Overlap Function dialog, click **Apply** and then **Close**.

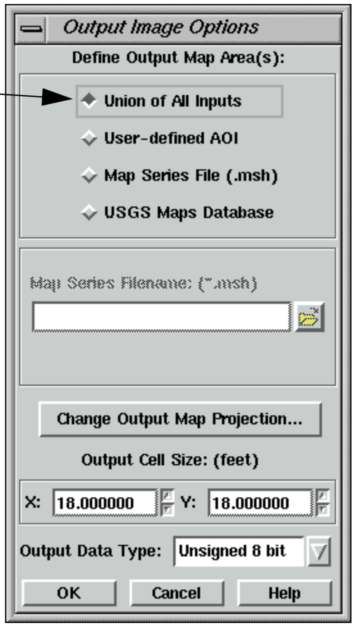
12. NOTE: The cutline viewer will automatically zoom to whatever is in the mosaic tool. In order to only show the overlapping area in the viewer, you need to click **scale canvas to fit selected objects**.

Define Output Images

1. In the Mosaic Tool viewer, click on the Output icon to activate the output mode.

2. In the Mosaic Tool viewer, click on the Output Image icon.

The Output Image Options dialog opens.



Make sure Union is selected

Output Image Options

Define Output Map Area(s):

● Union of All Inputs

○ User-defined AOI

○ Map Series File (.msh)

○ USGS Maps Database

Map Series Filename: (*.msh)

Output Cell Size: (feet)

X: 18.000000 Y: 18.000000

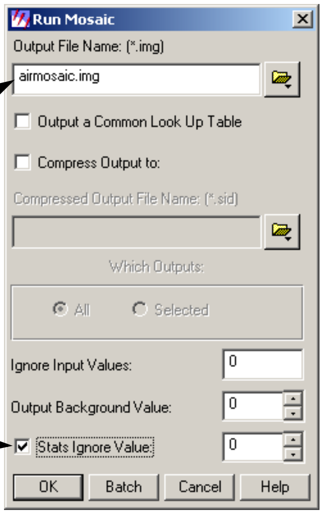
Output Data Type: Unsigned 8 bit

3. In the Output Image Options dialog under **Define Output Map Area(s)**, make sure that **Union of All Inputs** is selected and click **OK**.

Run the Mosaic

1. In the Mosaic Tool viewer, select **Process | Run Mosaic**.

   The Run Mosaic dialog opens.



Enter the name of the output file here

Click this check box to activate it

Run Mosaic

Output File Name: (*.img)

airmosaic.img

☐ Output a Common Look Up Table

☐ Compress Output to:

Compressed Output File Name: (*.sid)

Which Outputs:

● All ○ Selected

Ignore Input Values: 0

Output Background Value: 0

☑ Stats Ignore Value: 0

2. In the Run Mosaic dialog under **Output File Name**, enter **AirMosaic** in the directory of your choice, the press Return on your keyboard.

3. Click the **Stats Ignore Value** checkbox to activate it.

4. Click **OK** in the Run Mosaic dialog.

   A Job Status dialog displays, showing the progress of the function.

5. Click **OK** when the Job Status dialog reads that the function is 100% complete.

   NOTE: The Job Status dialog may close automatically, depending on your settings in **Session | Preferences | User Interface & Session | Keep Job Status Box**.

Display the Mosaic

1. Click on the Viewer icon in the ERDAS IMAGINE icon panel.

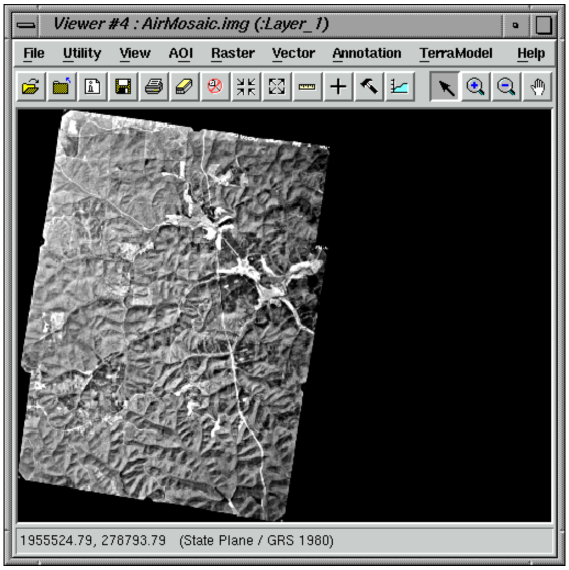
   A new Viewer displays.

2. Click the Open icon in the Viewer you just created.

   The Select Layer To Add dialog opens.

3. In the Select Layer To Add dialog under **Filename**, select **AirMosaic.img** from the directory in which you saved it, and then click **OK**.

   AirMosaic.img is displayed in the Viewer.



1955524.79, 278793.79 (State Plane / GRS 1980)

4. Compare **AirMosaic.img** to the original images (**air-photo-1.img** and **air-photo-2.img**).

Mosaic Using LANDSAT Images

In this section of the tour guide, you mosaic LANDSAT images of MSS and TM scenes.

The three files to be mosaicked are: **wasia1_mss.img, wasia2_mss.img, and wasia3_tm.img.**

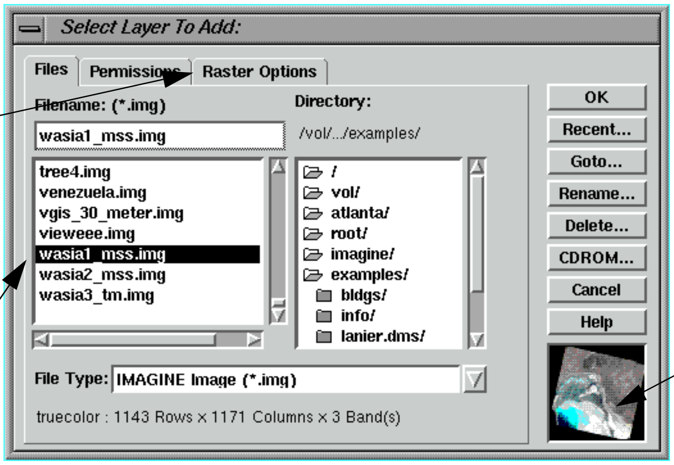
These data files are LANDSAT images of Kazakhstan, in the former USSR.

Display Input Images

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer, select **File | Open | Raster Layer** or click the Open icon.

   The Select Layer To Add dialog opens.



Select Layer To Add:
Files
Permissions
Raster Options
tree4.img
venezuela.img
vgis_30_meter.img
vieweee.img
wasia1_mss.img
wasia2_mss.img
wasia3_tm.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) wasia1_mss.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

truecolor : 1143 Rows x 1171 Columns x 3 Band(s)

Click on this file tab to view the raster options

Click here to select the file

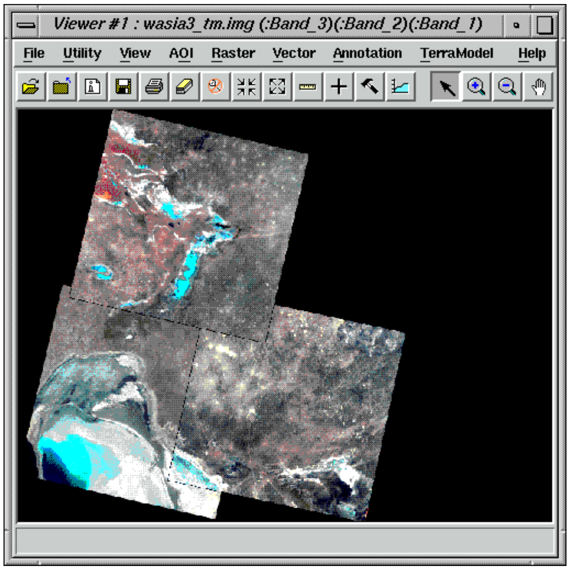
A preview of the image displays here

2. In the Select Layer To Add dialog under **Filename**, click on **wasia1_mss.img**
3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
4. Click the **Clear Display** option to disable it (this ensures that currently displayed files are not cleared in the Viewer).
5. Click the **Background Transparent** option to enable it.
6. Check to be sure that the **Fit to Frame** option is enabled.
7. Click **OK** in the Select Layer To Add dialog.

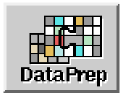
   The file **wasia1_mss.img** displays in the Viewer.

8. Repeat steps 1. through 7., opening **wasia2_mss.img** and **wasia3_tm.img** in the same Viewer you used for **wasia1_mss.img**.

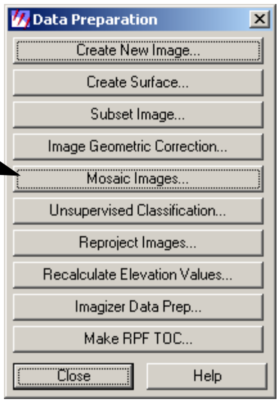
9. When finished, you should be able to see all three files in the Viewer:


Viewer #1 : wasia3_tm.img (:Band_3)(:Band_2)(:Band_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

10. In the ERDAS IMAGINE icon panel, click on the DataPrep icon.



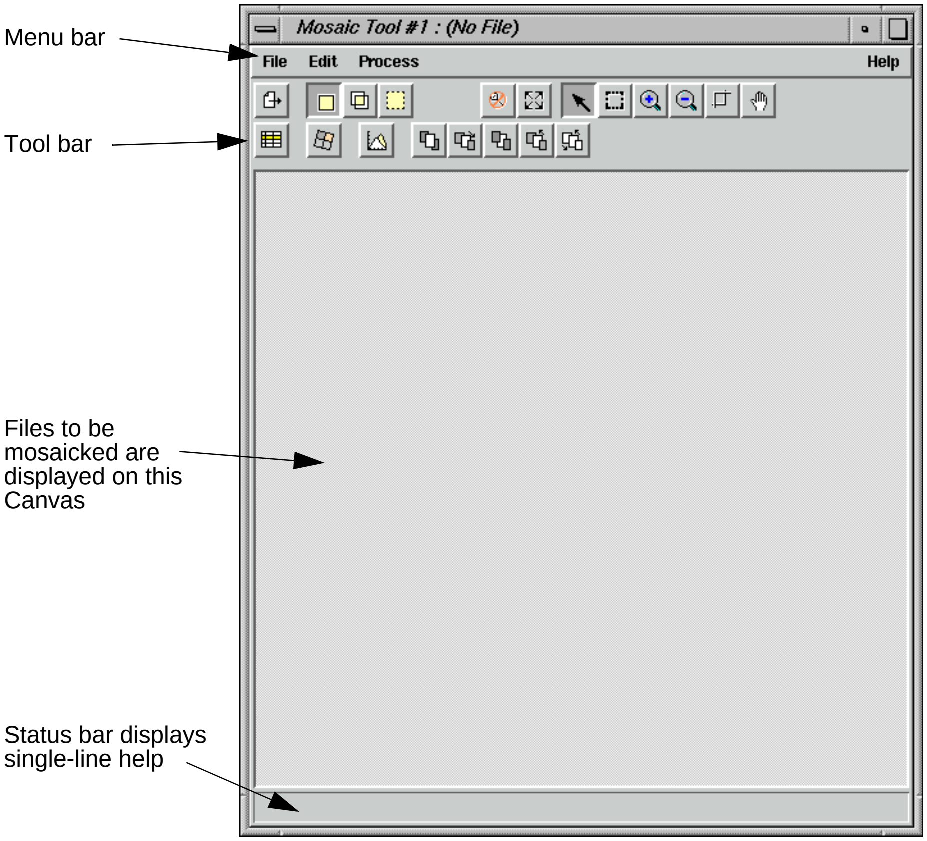
The **Data Preparation** menu displays.


Data Preparation
Create New Image...
Create Surface...
Subset Image...
Image Geometric Correction...
Mosaic Images...
Unsupervised Classification...
Reproject Images...
Recalculate Elevation Values...
Imagizer Data Prep...
Make RPF TOC...
Close
Help

Select Mosaic Images

11. In the **Data Preparation** menu, select **Mosaic Images**.

The Mosaic Tool viewer opens.


Menu bar
Tool bar
Files to be mosaicked are displayed on this Canvas
Status bar displays single-line help
Mosaic Tool #1 : (No File)
File
Edit
Process
Help

12. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Add Images for Mosaic

1. In the Mosaic Tool viewer, click on the Add Images icon.

The Add Images for Mosaic dialog opens.

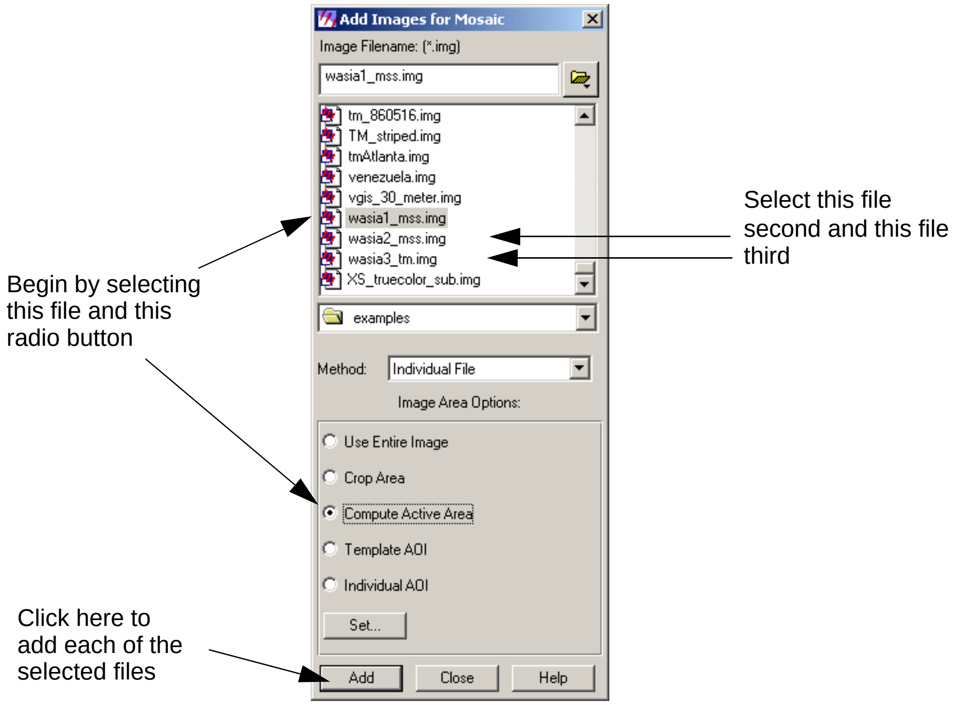

Add Images for Mosaic
tm_860516.img
TM_striped.img
tmAtlanta.img
venezuela.img
vgis_30_meter.img
wasia1_mss.img
wasia2_mss.img
wasia3_tm.img
XS_truecolor_sub.img
examples
Image Area Options:
Use Entire Image
Crop Area
Compute Active Area
Template AOI
Individual AOI
Set...
Add
Close
Help

Image Filename: (*.img) wasia1_mss.img

Method: Individual File

● Compute Active Area

Begin by selecting this file and this radio button

Select this file second and this file third

Click here to add each of the selected files

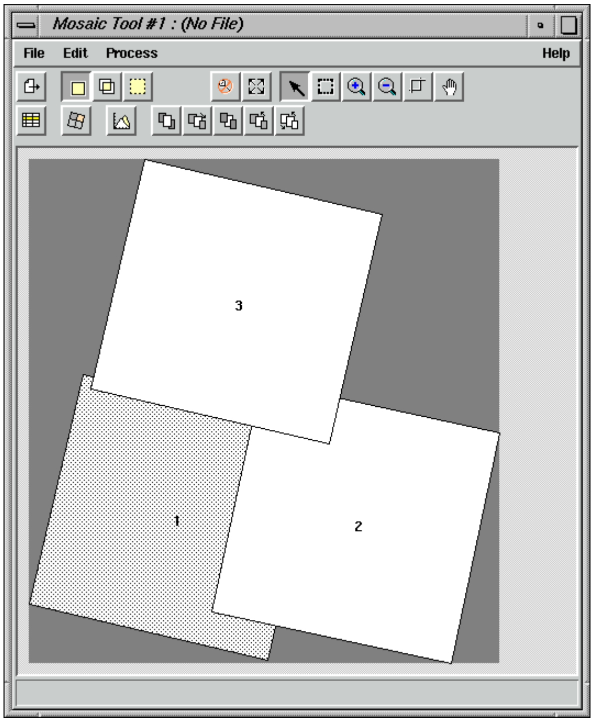
2. In the Add Images for Mosaic dialog under **Image Filename**, select **wasia1_mss.img**.

3. In the Add Images for Mosaic dialog, click the **Compute Active Area** button under **Image Area Options** to enable it, and then click **Add**.

   The file **wasia1_mss.img** displays as image number **1** in the canvas of the Mosaic Tool viewer.

4. Repeat steps 2. and 3., adding **wasia2_mss.img** and **wasia3_tm.img** to the canvas of the Mosaic Tool viewer. (Note that in step 3. the **Compute Active Area** button is already enabled.)

   When you are finished adding the three images, the Mosaic Tool viewer should look like the following:



5. Click **Close** in the Add Images for Mosaic dialog.

Stack Images

The images in the Mosaic Tool viewer are positioned as if they were regular photos that a person stacked by hand. One image intersects everything below it in the stack.

1. Experiment with the stacking order in the Mosaic Tool viewer by clicking on any or all of the images in the canvas and then clicking on any of the icons pictured below:

2. When you are finished experimenting with the stacking icons, click outside of the images in the canvas of the Mosaic Tool viewer to deselect the image(s).

Match Images

1. Click the Image Matching icon in the Mosaic Tool viewer.

The Matching Options dialog opens.

2. In the Matching Options dialog under **Matching Method**, select **Overlap Areas**.

3. Click **OK** in the Matching Options dialog.

4. Click on the Intersection icon in the Mosaic Tool tool bar.

5. In the Mosaic Tool tool bar, click the Function icon.

   The Set Overlap Function dialog opens.

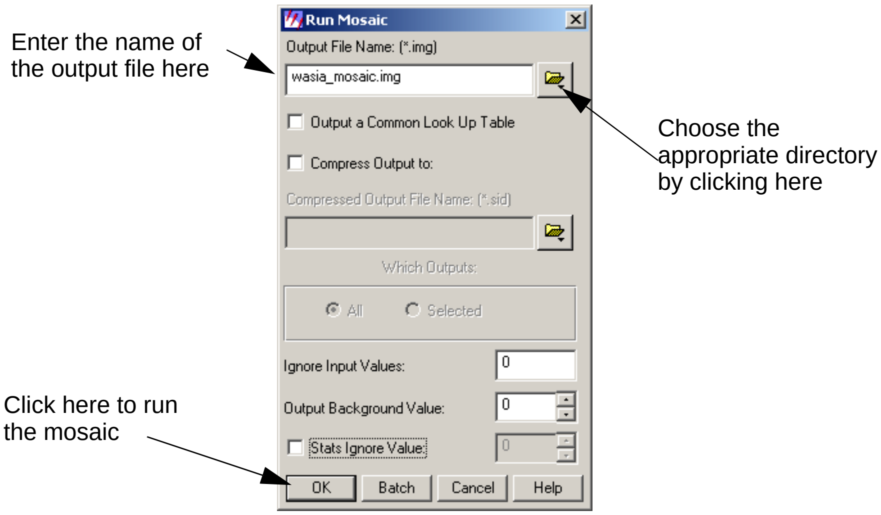
6. In the Set Overlap Function dialog under **Select Function**, select **Average** and then click **Apply**.

7. Click **Close** in the Set Overlap Function dialog.

Run the Mosaic

1. In the Mosaic Tool viewer, select **Process | Run Mosaic**.

   The Run Mosaic dialog opens.



Run Mosaic

Output File Name: (*.img) wasia_mosaic.img

☐ Output a Common Look Up Table

☐ Compress Output to:

Compressed Output File Name: (*.sid)

Which Outputs: ● All ○ Selected

Ignore Input Values: 0

Output Background Value: 0

☐ Stats Ignore Value: 0

Enter the name of the output file here

Choose the appropriate directory by clicking here

Click here to run the mosaic

2. In the Run Mosaic dialog under **Output File Name**, enter a name for the output file in the directory of your choice (e.g., **wasia_mosaic.img**).

3. Click **OK** in the Run Mosaic dialog.

   A Job Status dialog displays, stating the progress of the mosaic operation.

4. Click **OK** in the Job Status dialog when the mosaic operation is finished.

   NOTE: The mosaic operation can take up to ten minutes to run, based upon your hardware capabilities and the size of the files.

5. Select **File | Close** from the Mosaic Tool viewer to clear it from the screen.

   You are asked if you want to save the changes in the Mosaic Tool viewer. Save them if you like or click **No** to dismiss the dialog.

Display Output Image

1. Click the Viewer icon in the ERDAS IMAGINE icon panel to open a second Viewer.

2. In the Viewer you just opened, select **File | Open | Raster** or click the Open icon.

   The Select Layer To Add dialog opens.

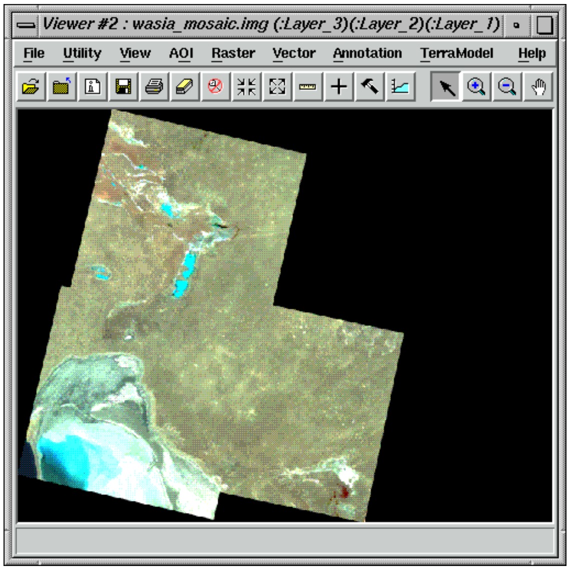
3. In the Select Layer To Add dialog under **Filename**, click on the file **wasia_mosaic.img** (or the output mosaic image you previously created). Remember to look in the directory in which you saved the file.

4. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

5. Check to be sure that the **Fit to Frame** option is enabled.

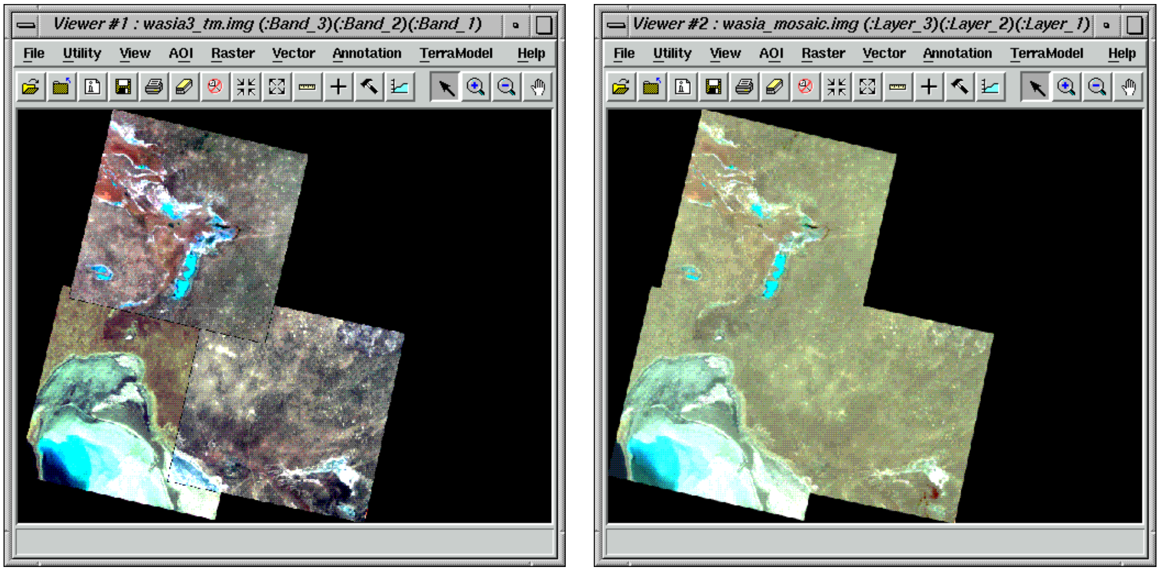
6. Click **OK** in the Select Layer To Add dialog.

   The output mosaic file displays in the Viewer.


Viewer #2 : wasia_mosaic.img (:Layer_3)(:Layer_2)(:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

7. In the ERDAS IMAGINE menu bar, select **Session | Tile Viewers** to position the Viewers side by side.

8. Compare the input mosaic images in the first Viewer to the output mosaic image in the second Viewer. You can resize the Viewers by dragging on any of their corners.


Viewer #1 : wasia3_tm.img (:Band_3)(:Band_2)(:Band_1)
File Utility View AOI Raster Vector Annotation TerraModel Help
Viewer #2 : wasia_mosaic.img (:Layer_3)(:Layer_2)(:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

Mosaic Color Balancing Using Color Infrared Aerial Photos

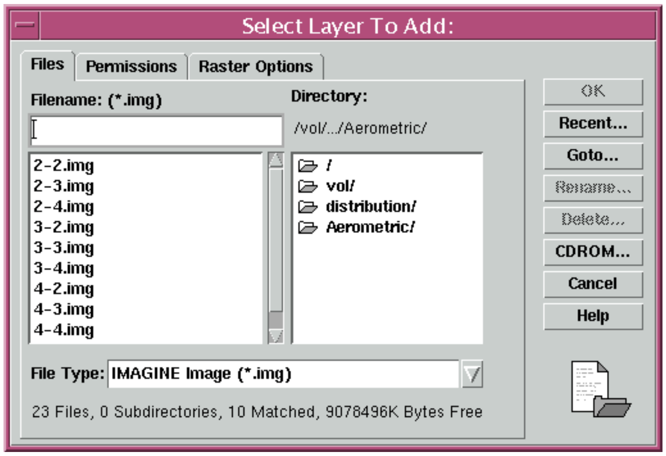
In this section of the tour guide, you mosaic color- infrared aerial photograph images of eastern Illinois, USA. The color red in the photos represents vegetation with deep reds representing healthy vegetation.

The nine files to be mosaicked are: **2-2.img, 2-3.img, 2-4.img, 3-2.img, 3-3.img, 3-4.img, 4-2.img, 4-3.img,** and **4-4.img**.

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer, select **File | Open | Raster Layer** or click the Open icon.

   The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
Filename: (*.img)
Directory:
2-2.img
2-3.img
2-4.img
3-2.img
3-3.img
3-4.img
4-2.img
4-3.img
4-4.img
/
vol/
distribution/
Aerometric/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
File Type:

Directory: /vol/.../Aerometric/

File Type: IMAGINE Image (*.img)

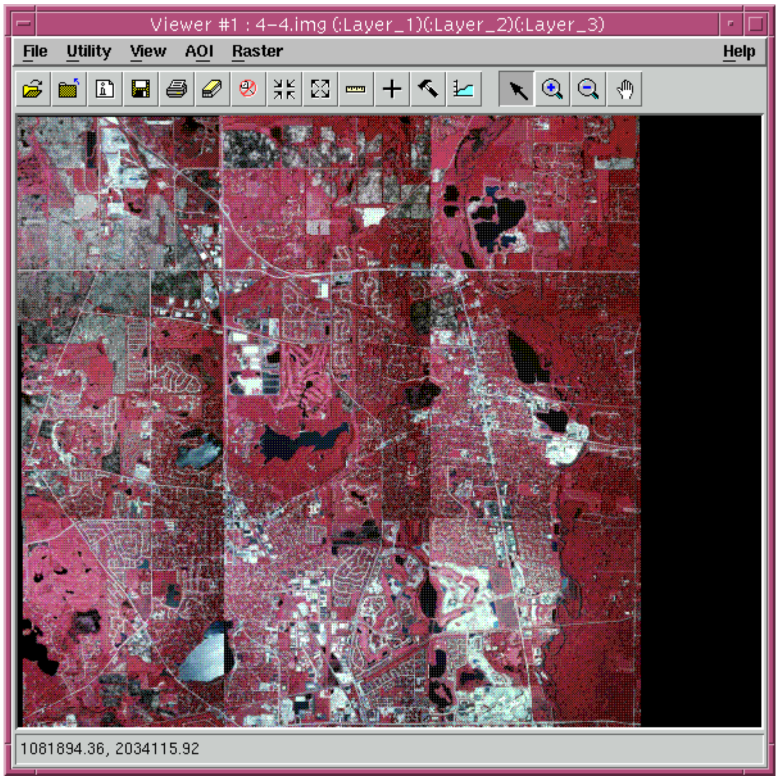
23 Files, 0 Subdirectories, 10 Matched, 9078496K Bytes Free

2. In the Select Layer To Add dialog under **Filename,** click on **2-2.img**.
3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
4. In the **Layers to Colors** box make sure **Red is 1**; **Green is 2**; and **Blue is 3**.
5. Click the **Clear Display** option to disable it (this ensures that currently displayed files are not cleared in the Viewer).
6. Click the **Fit to Frame** option to enable it.
7. Click the **Background Transparent** option to enable it.
8. Click **OK** in the Select Layer To Add dialog.

   The file **2-2.img** displays in the Viewer.

9. Repeat steps 1. through 8., opening **2-3.img**, **2-4.img**, **3-2.img**, **3-3.img**, **3-4.img**, **4-2.img**, **4-3.img**, and **4-4.img** in the same Viewer you used for **2-2.img**.

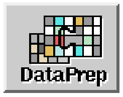
   When finished, you should be able to see all nine files in the Viewer.



1081894.36, 2034115.92

You will see patterned color differences in some of the images. Some spots in some of the photos will appear brighter or darker than the rest of the image. In order to adjust these color differences, you will use the **Mosaic Color Balancing** option in the **Mosaic Tool**.

10. In the ERDAS IMAGINE icon panel, click on the **DataPrep** icon.



The **Data Preparation** menu displays.

11. In the **Data Preparation** menu, select **Mosaic Images**.

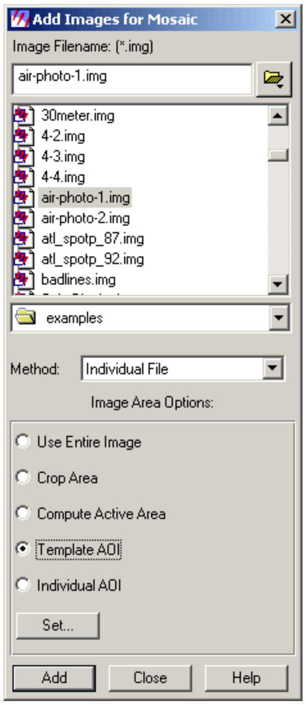
    The Mosaic Tool viewer opens.

12. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Add Images for Mosaic

1. In the Mosaic Tool viewer, click on the Add Images icon.

The Add Images for Mosaic dialog opens.



Add Images for Mosaic

Image Filename: (*.img) air-photo-1.img

Method: Individual File

Image Area Options:

○ Use Entire Image
○ Crop Area
○ Compute Active Area
● Template AOI
○ Individual AOI

30meter.img
4-2.img
4-3.img
4-4.img
air-photo-1.img
air-photo-2.img
atl_spotp_87.img
atl_spotp_92.img
badlines.img
examples
Set...
Add
Close
Help

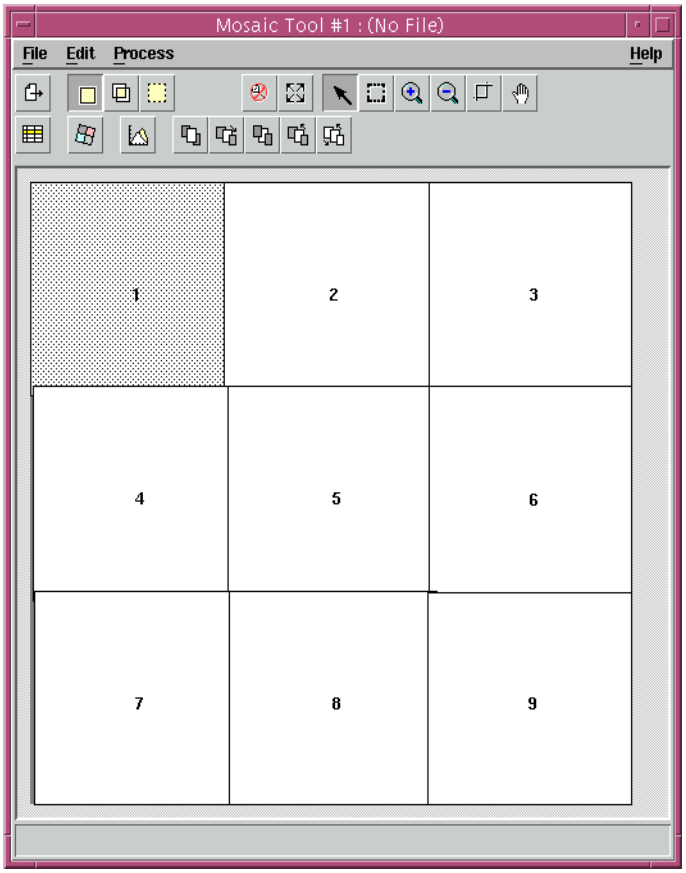
2. In the Add Images for Mosaic dialog under **Image Filename**, select **2-2.img**.

3. In the Add Images for Mosaic dialog, make sure the **Use Entire Image** button is selected, and then click **Add**.

   The file **2-2.img** displays as image number **1** in the canvas of the Mosaic Tool viewer.

4. Repeat steps 2. and 3., adding **2-3.img**, **2-4.img**, **3-2.img**, **3-3.img**, **3-4.img**, **4-2.img**, **4-3.img**, and **4-4.img** to the canvas of the Mosaic Tool viewer.

   When you are finished adding the nine images, the Mosaic Tool viewer should look like the following:

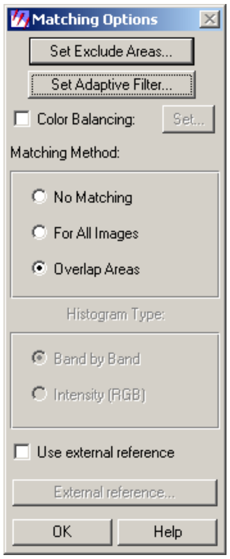

Mosaic Tool #1 : (No File)
File
Edit
Process
Help
1
2
3
4
5
6
7
8
9

5. Click **Close** in the Add Images for Mosaic dialog.

Set Exclude Areas

1. Click the Image Matching icon in the Mosaic Tool viewer.

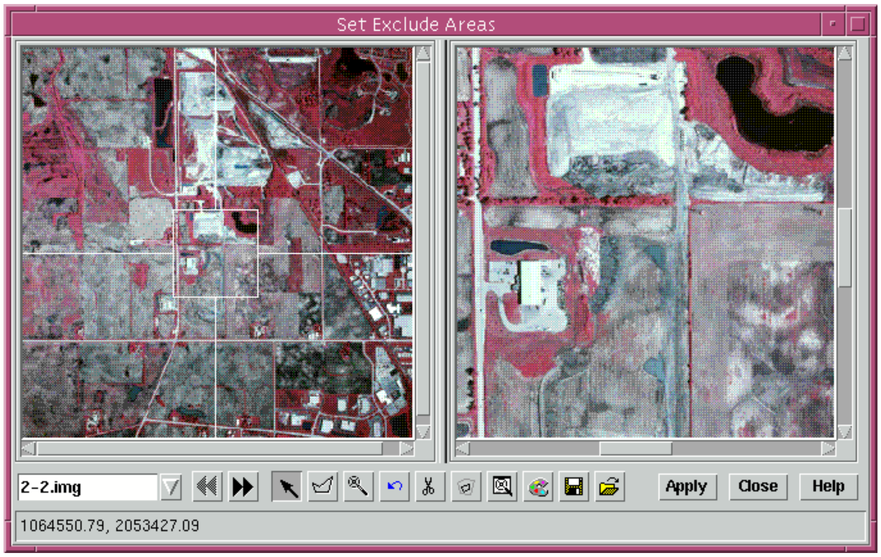
The Matching Options dialog opens.


Matching Options
Set Exclude Areas...
Set Adaptive Filter...
☐ Color Balancing:
Set...
Matching Method:
○ No Matching
○ For All Images
● Overlap Areas
Histogram Type:
● Band by Band
○ Intensity (RGB)
☐ Use external reference
External reference...
OK
Help

2. Select **Color Balancing**, and under **Matching Method** choose **For All Images**.

3. Click on **Set Exclude Areas** at the top of the Matching Options dialog.

   The Set Exclude Areas dialog opens.


Set Exclude Areas
2-2.img
Apply
Close
Help
1064550.79, 2053427.09

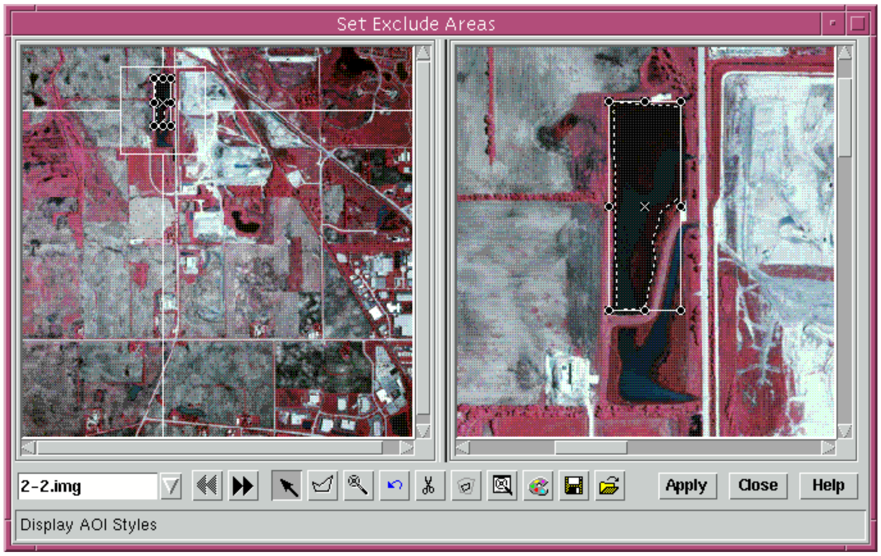
The overview shows the full extent of the image, and the detail view allows you to zoom in on specific areas to create AOIs to be excluded from the Color Balancing process. Use the link cursor to isolate AOIs.

4. Image **2-2.img** will be the first image displayed in the viewers. Left hold your mouse and drag the link cursor to the dark body of water in the upper left of the center of the image.

   In the **Detail View**, the area is magnified so you can create specific AOIs to exclude from the Color Balancing process.

5. Click the Create Polygon AOI icon.

6. Your cursor becomes a cross hair when inside a view. Use the **Detail View** to create polygons around the dark water bodies.



7. When you have finished, use the link cursor to identify any other areas in the image that need to be excluded such as other dark bodies of water, isolated bright urban areas, or bright areas of sun glint.

8. Create a polygon around each of the areas you wish to exclude from color balancing, and click **Apply**.

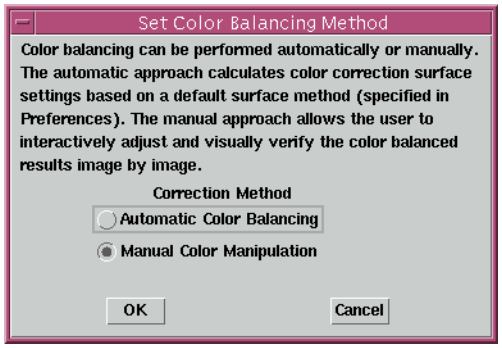
9. Use the arrow keys or the drop down box to select the next image, **2-3.img**

   Continue identifying AOIs to exclude through all nine images. Click **Apply** after excluding areas in each image. Click **Close** in the Set Exclude Areas dialog when finished.

Mosaic Color Balancing

1. Click the **Set** button in the Matching Options dialog.

The **Set Color Balancing Method** dialog opens.



Set Color Balancing Method

Color balancing can be performed automatically or manually. The automatic approach calculates color correction surface settings based on a default surface method (specified in Preferences). The manual approach allows the user to interactively adjust and visually verify the color balanced results image by image.

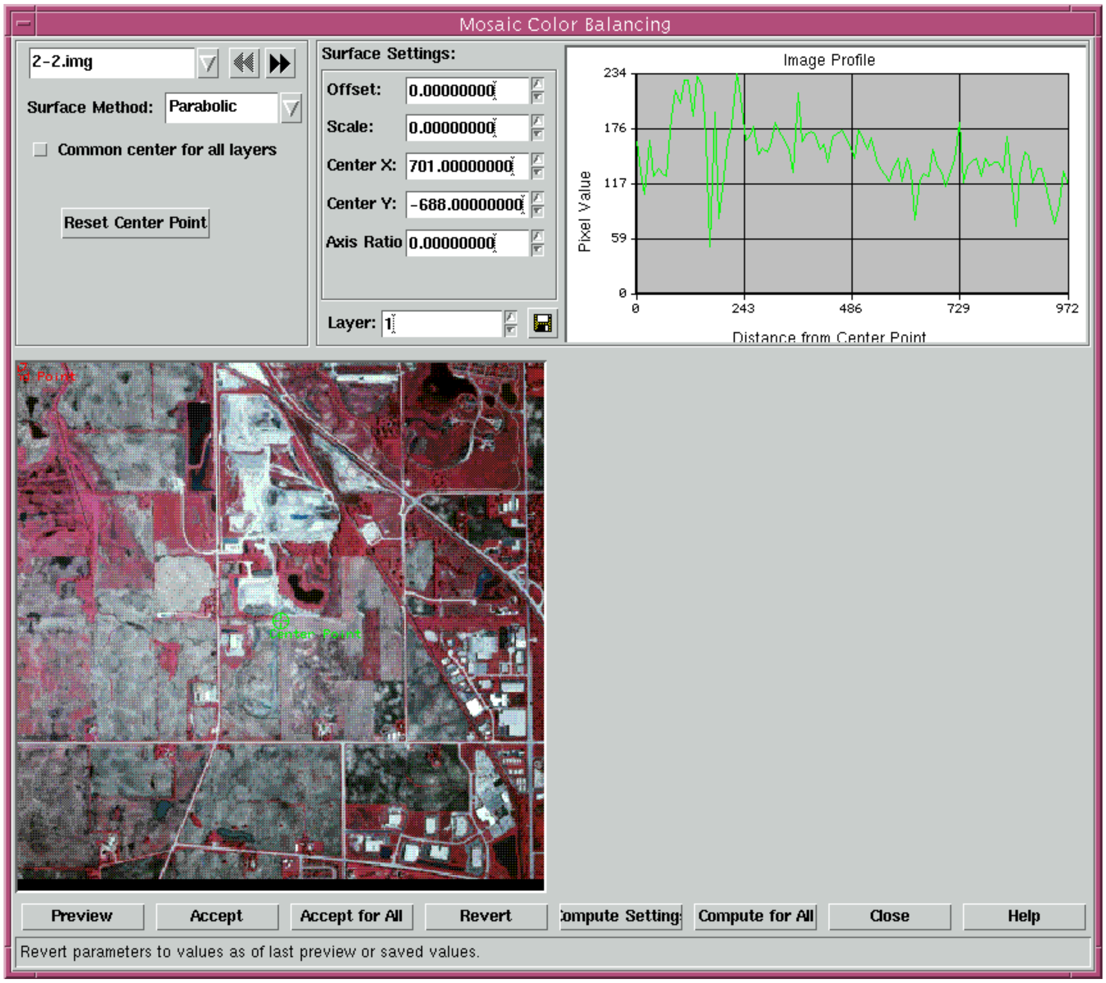
Correction Method

○ Automatic Color Balancing

● Manual Color Manipulation

2. Choose **Manual Color Manipulation** and click **OK.**.

The **Mosaic Color Balancing** dialog opens.


Mosaic Color Balancing
Surface Settings:
Image Profile
Pixel Value
234
176
117
59
0
0
243
486
729
972
Distance from Center Point
Center Point
Preview
Accept
Accept for All
Revert
ompute Setting
Compute for All
Close
Help
Reset Center Point

2-2.img

Surface Method: Parabolic

☐ Common center for all layers

Offset: 0.00000000

Scale: 0.00000000

Center X: 701.00000000

Center Y: -688.00000000

Axis Ratio 0.00000000

Layer: 1

Revert parameters to values as of last preview or saved values.

3. The first image, **2-2.img**, is displayed. If **2-2.img** did not need color balancing, you could skip it, and use the **Arrow Keys** or the Drop Down Menu in the top left corner to choose another image.

4. Choose the surface method you want to use in order to color balance the image. You may choose from **Parabolic**, **Conic**, **Linear**, or **Exponential**.

   Unless you are using **Linear** as your surface method, you can click on **Common center for all layers** in order to use the same center point in each layer of the image.

5. For image **2-2.img**, select **Conic**, and enable **Common center for all layers**.

6. Click **Compute Setting** and click on **Preview**. The preview viewer displays the color balanced image.

7. Click **Accept** after previewing the color balanced image to accept it for mosaicking.

The computed settings are based on the surface method chosen.The **Image Profile** box at the top right of the **Mosaic Color Balancing** dialog depicts the chosen surface method as a red line plotted against a profile of the pixel values shown as a green line between the center point and end point in the image. Results are best when the red line of the surface method matches the general trend of the green line of the image profile.

8. Repeat steps 3. through 6. for images **2-3.img**, **2-4.img**, **3-2.img**, **3-3.img**, **3-4.img**, **4-2.img**, **4-3.img**, and **4-4.img**.
9. Click **Close** in the Mosaic Color Balancing dialog.

Match Images

1. Click the Image Matching icon in the Mosaic Tool viewer.

   The Matching Options dialog opens.

2. In the Matching Options dialog under **Matching Method**, select **For All Images**
3. Select **Band by Band** under Histogram Type.
4. Click **OK** in the Matching Options dialog.
5. Click on the Intersection icon in the Mosaic Tool tool bar.
6. Click on the Set Default Cutlines icon.

   An attention box will appear. Click **Yes** and continue.

7. In the Mosaic Tool tool bar, click the Function icon.

   The Set Overlap Function dialog opens.

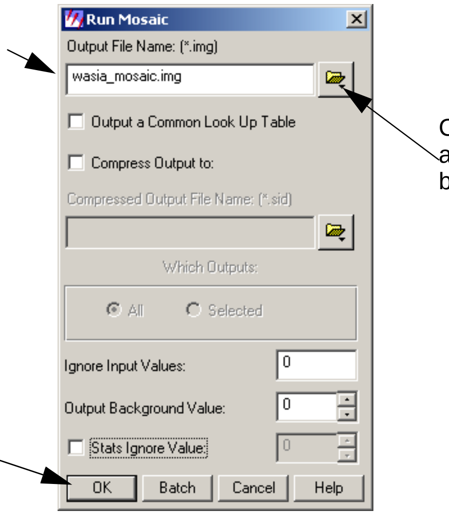
8. In the Set Overlap Function dialog under **Intersection Type**, select **Cutline Exists** and **Cut/Feather** then click **Apply**.

9. Click **Close** in the Set Overlap Function dialog.

Run The Mosaic

1. In the Mosaic Tool viewer, select **Process | Run Mosaic**.

   The Run Mosaic dialog opens.



Output File Name: (*.img) wasia_mosaic.img
☐ Output a Common Look Up Table
☐ Compress Output to:
Compressed Output File Name: (*.sid)
Which Outputs: ● All ○ Selected
Ignore Input Values: 0
Output Background Value: 0
☐ Stats Ignore Value: 0

Enter the name of the output file here

Choose the appropriate directory by clicking here

Click here to run the mosaic

2. In the Run Mosaic dialog under **Output File Name**, enter a name for the output file in the directory of your choice (e.g., **color_infrared.img**).

3. Click **OK** in the Run Mosaic dialog.

   A Job Status dialog displays, stating the progress of the mosaic operation.

4. Click **OK** in the Job Status dialog when the mosaic operation is finished.

   NOTE: The mosaic operation can take up to ten minutes to run, based upon your hardware capabilities and the size of the files.

5. Select **File | Close** from the Mosaic Tool viewer to clear it from the screen.

Display Output Image

1. Click on the Viewer icon in the ERDAS IMAGINE icon panel.

   A new Viewer displays.

2. Click the Open icon in the Viewer you just created.

The Select Layer To Add dialog opens.

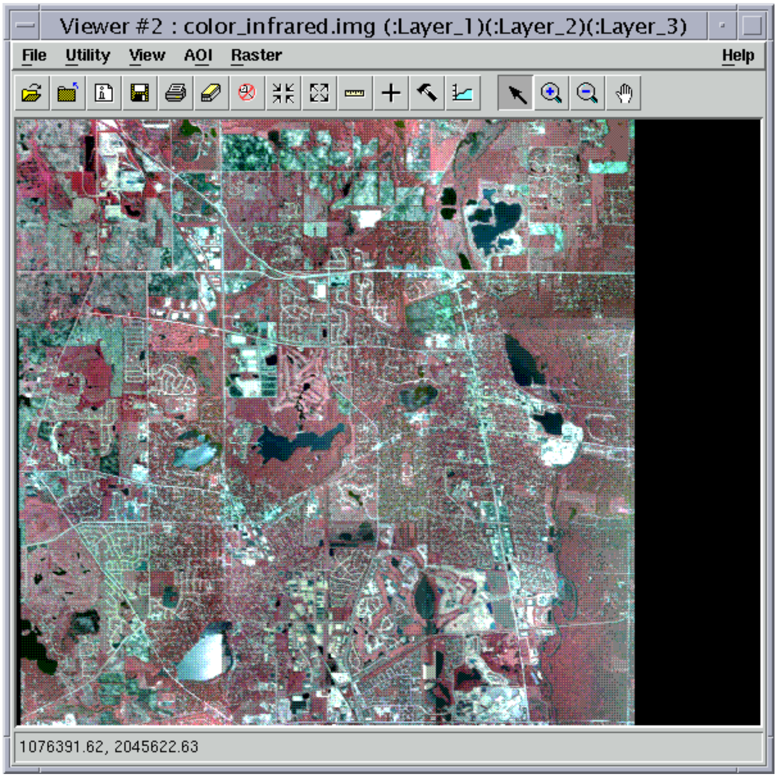
3. In the Select Layer To Add dialog under **Filename**, select **color_infrared.img** from the directory in which you saved it, and then click **OK**.

4. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

5. Check to be sure the **Fit to Frame** option is enabled.

6. Click **OK** in the Select Layer To Add dialog.

The mosaicked output image displays in the viewer.


Viewer #2 : color_infrared.img (:Layer_1)(:Layer_2)(:Layer_3)
File
Utility
View
AOI
Raster
Help

1076391.62, 2045622.63

CHAPTER FIFTEEN

Viewshed Analysis

Introduction

One of the many tasks you can perform using IMAGINE Advantage is Viewshed Analysis. This tour guide describes how to use this analysis tool.

Viewshed Analysis allows you to position an observer on a DEM in a Viewer and determine the visible areas within the terrain. You can adjust the observer's height either above ground level or above sea level and set the visible range.

This tool is useful for planning the location and height of towers used for observation or communications. It might also be used to determine areas that lie within poor reception of standard broadcast towers and are thus potential cable markets.

In this tour guide, you can learn how to:

- start an Image Drape viewer
- start the Viewshed tool
- work with multiple observers
- query Viewshed data and layers

Approximate completion time for this tour guide is 15 minutes.

Create a Viewshed

In this exercise, you create a viewshed, and analyze the terrain within it.

ERDAS IMAGINE must be running with a Viewer open.

1. Click the Open icon in the Viewer (or select **File | Open | Raster Layer**).
2. In the Select Layer To Add dialog, navigate to the <IMAGINE_HOME>/examples directory.
3. Select the file **eldodem.img**, then click the **Raster Options** tab.
4. In the **Raster Options** tab, make sure that the **Fit to Frame** checkbox is active.
5. Click **OK** in the Select Layer To Add dialog.

Add the Raster Image

1. Click the Open icon again, and navigate to the <IMAGINE_HOME>/examples directory.
2. Select the file **eldoatm.img**, then click the **Raster Options** tab.
3. In the **Raster Options** tab, make sure that the **Clear Display** checkbox is not active.
4. Click **OK** in the Select Layer to Add dialog.

   Both files are displayed in the Viewer.

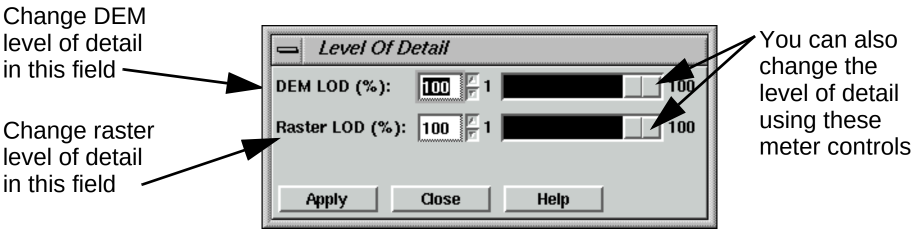
Start an Image Drape Viewer and Set the Level of Detail

1. From the menu bar of the Viewer displaying **eldodem.img**, select **Utility | Image Drape**.

   The Image Drape Viewer opens displaying **eldodem.img**, which supplies terrain relief, and **eldoatm.img**, which supplies the color. Position the Image Drape Viewer so that it does not cover the first Viewer you opened. Now, you can set the level of detail.

2. Select **View | LOD Control** from the Image Drape viewer menu bar.

   The Level Of Detail dialog opens.



DEM LOD (%): 100

Raster LOD (%): 100

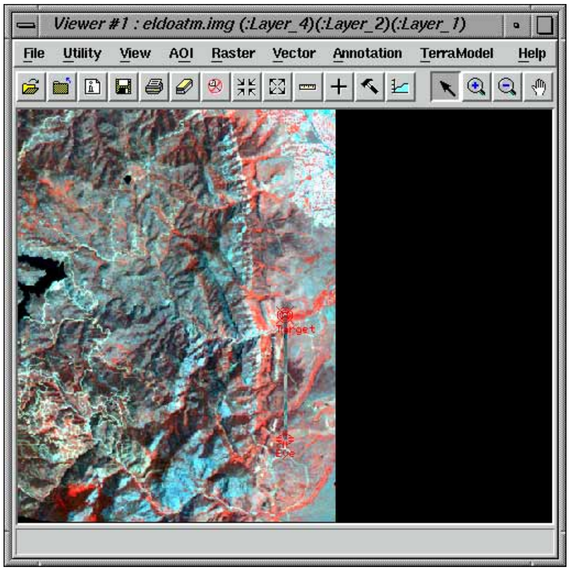
Change DEM level of detail in this field

Change raster level of detail in this field

You can also change the level of detail using these meter controls

3. In the field next to **DEM LOD (%)**, enter **100**, and press Return on your keyboard.
4. Click **Apply** to increase the level of detail in the Image Drape viewer.
5. Click **Close** to dismiss the Level Of Detail dialog.

   Your two Viewers now look like the following:


Viewer #1 : eldoatm.img (:Layer_4)(:Layer_2)(:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
Target
Eye

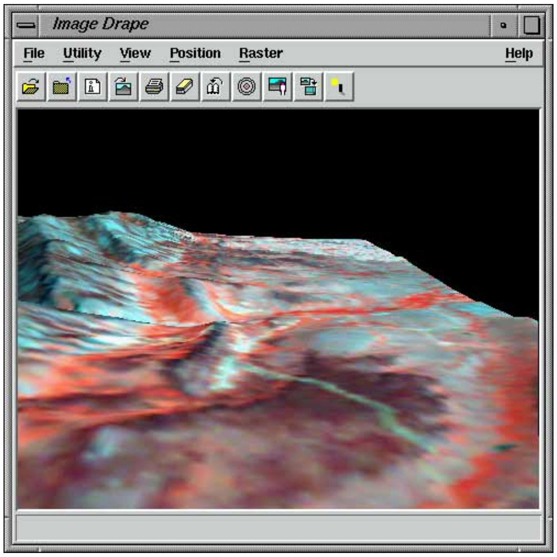
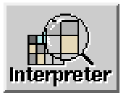


Image Drape
File
Utility
View
Position
Raster
Help

Start the Viewshed Analysis Tool

1. Click the Interpreter icon on the ERDAS IMAGINE icon bar.


Interpreter

The **Image Interpreter** menu opens.

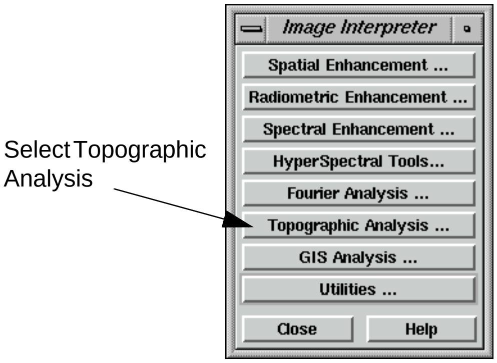
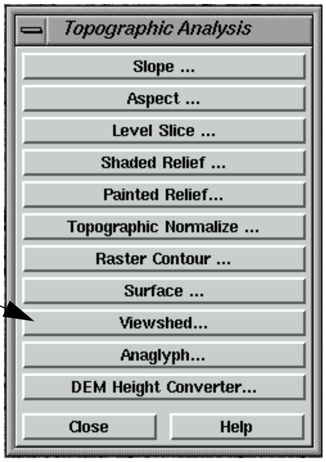


Image Interpreter
Spatial Enhancement ...
Radiometric Enhancement ...
Spectral Enhancement ...
HyperSpectral Tools...
Fourier Analysis ...
Topographic Analysis ...
GIS Analysis ...
Utilities ...
Close
Help

Select Topographic Analysis

2. From the **Image Interpreter** menu, select **Topographic Analysis.**

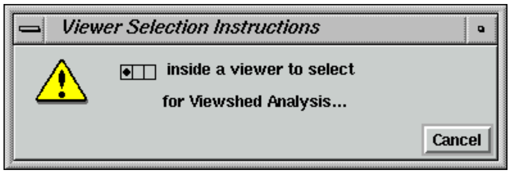
The **Topo Analysis** menu opens.


Topographic Analysis
Slope ...
Aspect ...
Level Slice ...
Shaded Relief ...
Painted Relief...
Topographic Normalize ...
Raster Contour ...
Surface ...
Viewshed...
Anaglyph...
DEM Height Converter...
Close
Help

Click Viewshed

3. From the **Topo Analysis** menu, select **Viewshed**.

   A Viewer Selection Instructions dialog opens.

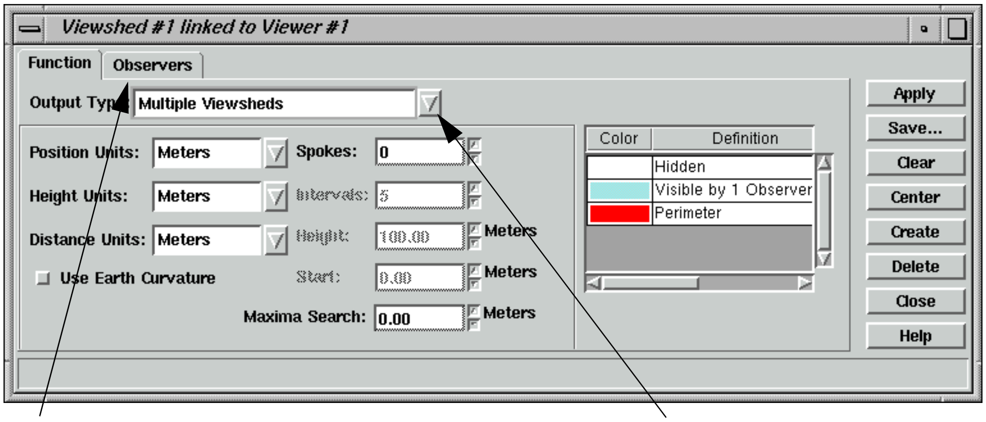

Viewer Selection Instructions
Cancel

inside a viewer to select
for Viewshed Analysis...

4. Click in the Viewer containing **eldodem.img** and **eldoatm.img**.

   The Viewshed dialog opens.

5. At this time, click **Close** on both the **Image Interpreter** menu and the **Topo Analysis** menu to remove them from your display.


Viewshed #1 linked to Viewer #1
Function
Observers
Color
Definition
Apply
Save...
Clear
Center
Create
Delete
Close
Help

Output Type: Multiple Viewsheds
Position Units: Meters
Height Units: Meters
Distance Units: Meters
☐ Use Earth Curvature
Spokes: 0
Intervals: 5
Height: 100.00 Meters
Start: 0.00 Meters
Maxima Search: 0.00 Meters

Color	Definition
☐	Hidden
🟦	Visible by 1 Observer
🟥	Perimeter

Click the Observers tab to add an observer

Select Multiple Viewsheds

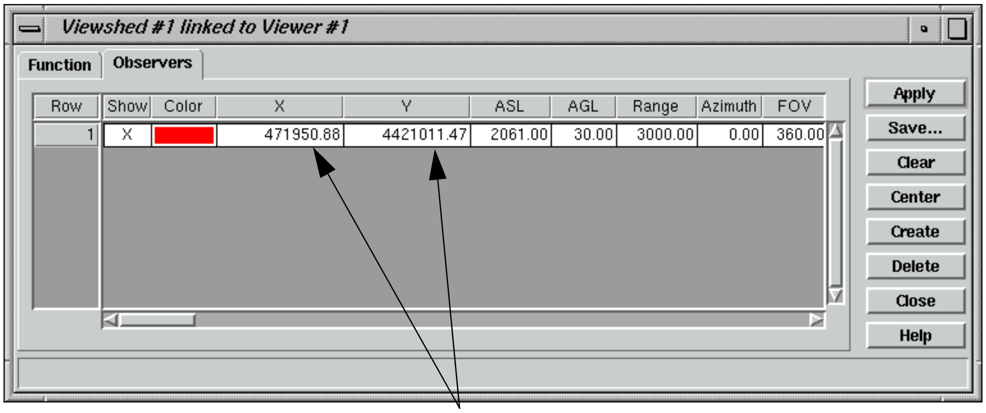
An observer marker is automatically placed in the center of the Viewer containing **eldodem.img**.

6. In the **Function** tab of the Viewshed dialog, click the popup list next to **Output Type** to select **Multiple Viewsheds**.

Add First Observer

1. Click the **Observers** tab in the Viewshed dialog.

   The **Observers** tab of the CellArray displays.


Viewshed #1 linked to Viewer #1
Function
Observers
Apply
Save...
Clear
Center
Create
Delete
Close
Help

Row	Show	Color	X	Y	ASL	AGL	Range	Azimuth	FOV
1	X	🟥	471950.88	4421011.47	2061.00	30.00	3000.00	0.00	360.00

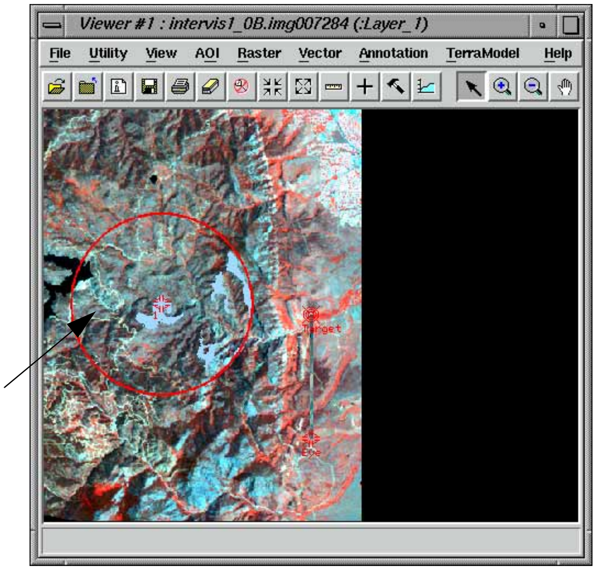
Specify the observer’s position in these columns

2. Click in the cell of the **X** column to enter **471950.88**, then press Return on your keyboard.

3. Click in the cell of the **Y** column to enter **4421011.47**, then press Return on your keyboard.

4. Click **Apply** in the Viewshed dialog.

   The viewshed layer is generated and displays in the Viewer.

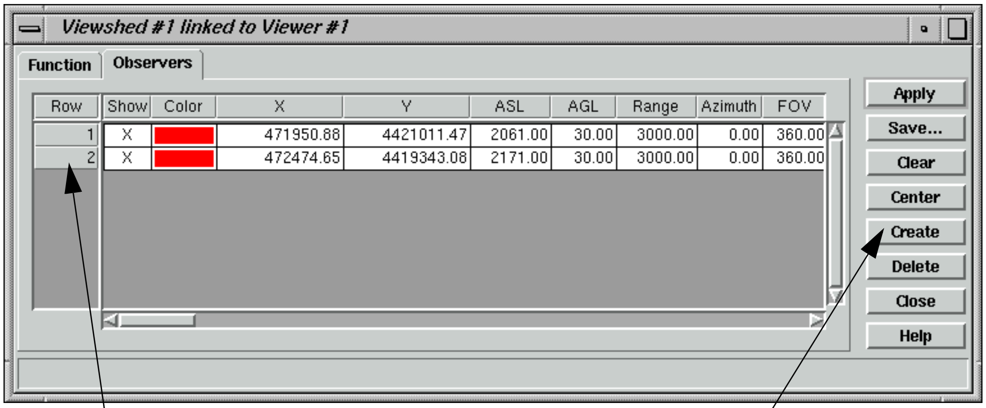

Viewer #1 : intervis1_0B.img007284 (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
Target
Eye

The area of the viewshed is marked by a circle

Add Another Observer

1. Click **Create** in the Viewshed dialog.

   A new observer is added to the CellArray.


Viewshed #1 linked to Viewer #1
Function
Observers
Apply
Save...
Clear
Center
Create
Delete
Close
Help

Row	Show	Color	X	Y	ASL	AGL	Range	Azimuth	FOV
1	X	🟥	471950.88	4421011.47	2061.00	30.00	3000.00	0.00	360.00
2	X	🟥	472474.65	4419343.08	2171.00	30.00	3000.00	0.00	360.00

The second observer is added in Row 2

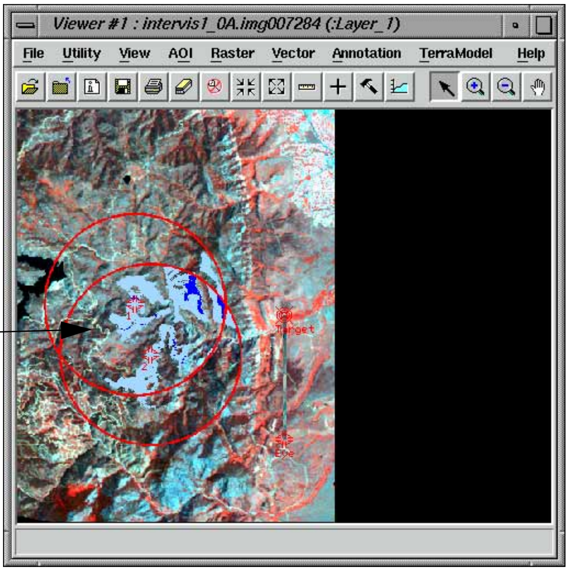
Click Create

2. Click in the cell of the second observer’s **X** column to enter **472474.65**, then press Return on your keyboard.

3. Click in the cell of the second observer’s **Y** column to enter **4419343.08**, then press Return on your keyboard.

4. Click **Apply** in the Viewshed dialog.

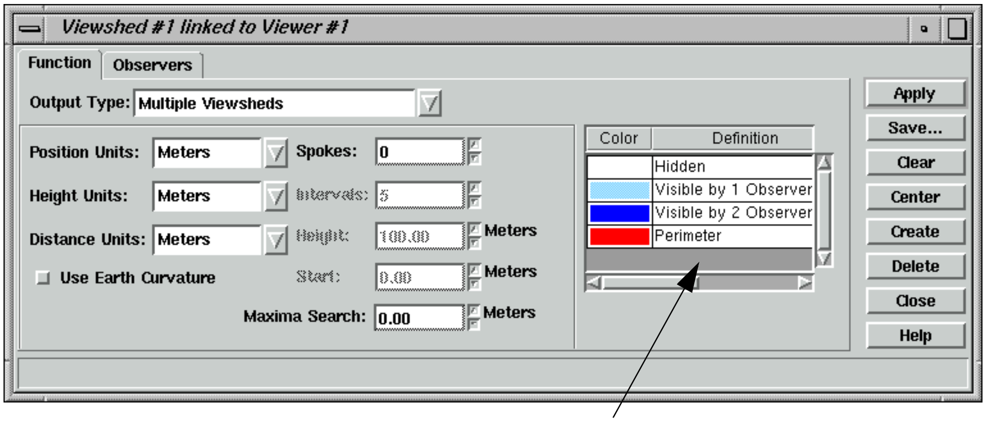
   The second viewshed layer is generated and displays in the Viewer.


Viewer #1 : intervis1_0A.img007284 (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help
Target
Eye

Portions of the two viewsheds intersect, evidenced by the overlap of the red circles

5. Click the **Function** tab in Viewshed dialog to view the legend.

   The **Function** tab opens, displaying the legend of the viewshed.


Viewshed #1 linked to Viewer #1
Function
Observers
Apply
Save...
Clear
Center
Create
Delete
Close
Help

Output Type: Multiple Viewsheds

Position Units: Meters

Height Units: Meters

Distance Units: Meters

☐ Use Earth Curvature

Spokes: 0

Intervals: 5

Height: 100.00 Meters

Start: 0.00 Meters

Maxima Search: 0.00 Meters

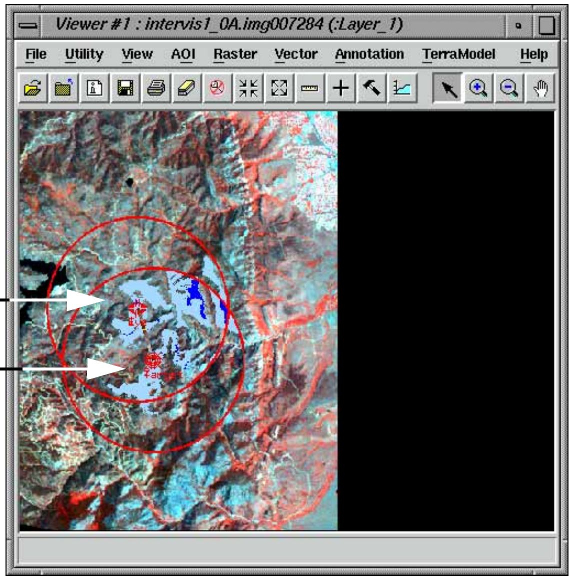
Color	Definition
⬜	Hidden
🟦	Visible by 1 Observer
🟦	Visible by 2 Observer
🟥	Perimeter

The legend displays here, in the Function tab

There are two basic kinds of output. The Viewshed outputs provide a binary analysis of visibility within the specified range. In other words, the image is color-coded to show only visible or hidden areas. The Height outputs provide a color-coded map of the invisible areas indicating the amount of change in observer height required to see a given zone.

Link the Viewers and Set Eye and Target Positions

1. Using your mouse, move the **Eye** of the Positioning tool on top of **Observer 1**.
2. Move the **Target** on top of **Observer 2** in the Viewer.


Viewer #1 : intervis1_0A.img007284 (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

The Eye of the position tool is placed on top of Observer 1

The Target of the position tool is placed on top of Observer 2

As you move the Positioning tool, the 3D image in the Image Drape viewer is updated. **Observer 1** is now looking at the location of **Observer 2**. The 3D image is positioned so that the target is centered in the Image Drape viewer.

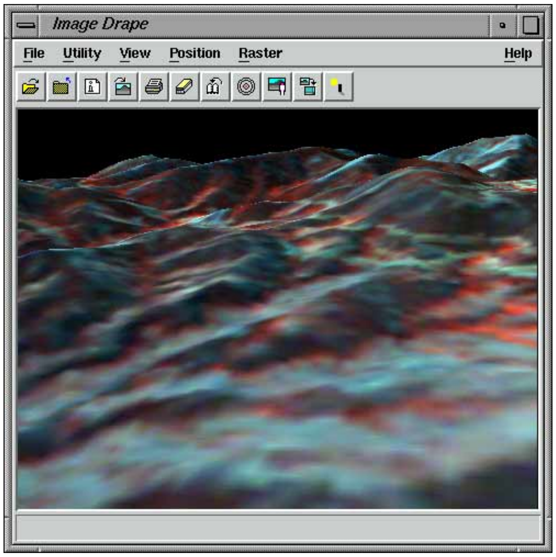


Image Drape
File
Utility
View
Position
Raster
Help

3. Switch the **Eye** and **Target** of the Positioning tool in the Viewer.

   Observer 2 is now looking at the location of **Observer 1**.

   Again, the 3D image rotates to match the **Eye** and **Target** positions of the Viewer.

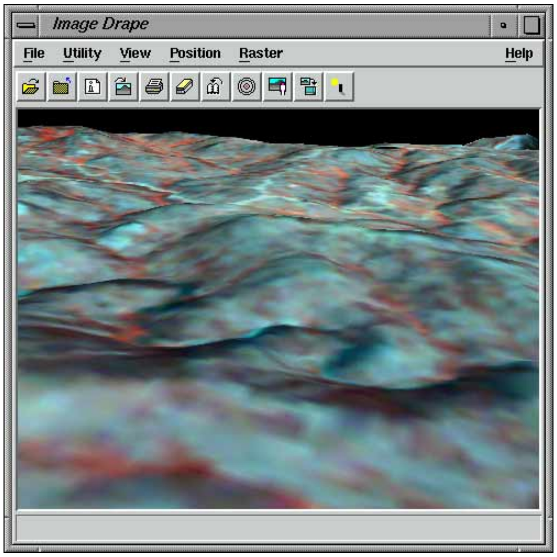


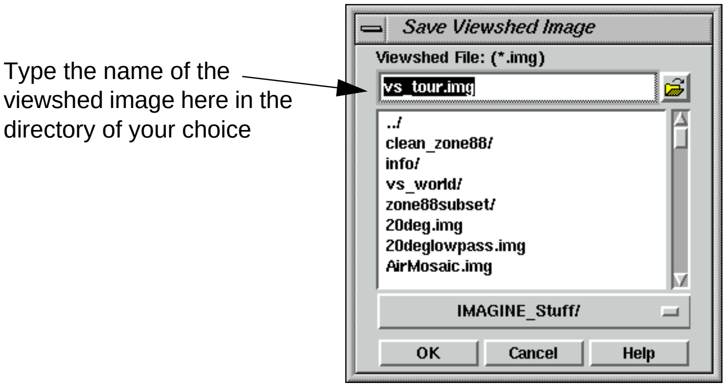
Image Drape
File Utility View Position Raster Help

Save the Viewshed

What can we do with a saved viewshed? For example, the saved viewshed can be used to create a map composition.

1. In the Viewshed dialog, click **Save**.

   The Save Viewshed Image dialog opens.


Type the name of the viewshed image here in the directory of your choice
Save Viewshed Image
Viewshed File: (*.img)
vs_tour.img
../
clean_zone88/
info/
vs_world/
zone88subset/
20deg.img
20deglowpass.img
AirMosaic.img
IMAGINE_Stuff/
OK
Cancel
Help

2. In a directory where you have write permission, type **vs_tour.img** in the **Viewshed File** window.

3. Click **OK** to dismiss the Save Viewshed Image dialog.

A Viewshed Analysis progress meter appears while the image is saved.

4. Click **Close** to dismiss the Viewshed dialog.

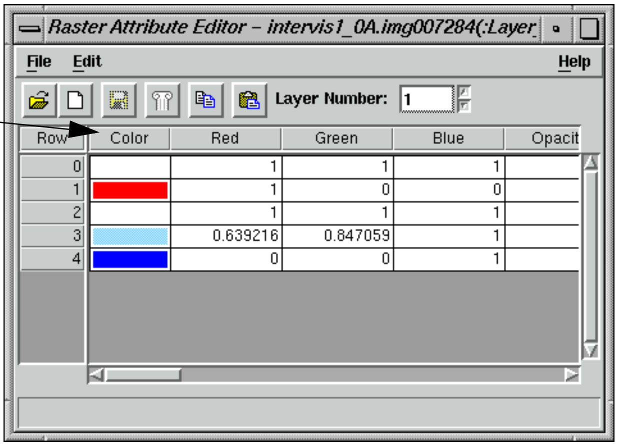
Query Viewshed Data

In this section, use the Raster Attribute Editor to query the viewshed layer in the Viewer.

Create Class Names for Viewshed Regions

1. Select **Raster | Attributes** from the Viewer menu bar.

The Raster Attribute Editor opens.



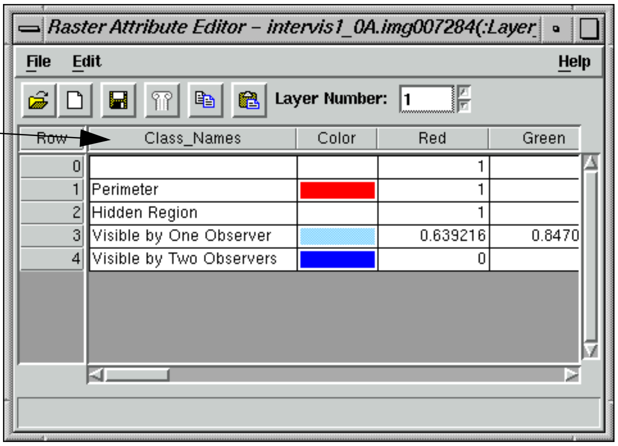
Row	Color	Red	Green	Blue	Opacit
0	⬜	1	1	1	
1	🟥	1	0	0	
2	⬜	1	1	1	
3	🟦	0.639216	0.847059	1	
4	🟦	0	0	1	

Add a column for class names

2. Select **Edit | Add Class Names** from the Raster Attribute Editor menu bar.

A new column is added to the front of the Raster Attributes CellArray.

3. In **Row 1** of the **Class Names** column, enter **Perimeter**, and press Return on your keyboard.
4. In **Row 2**, enter **Hidden Region**, and press Return on your keyboard.
5. In **Row 3**, enter **Visible by One Observer**, and press Return on your keyboard.
6. In **Row 4**, enter **Visible by Two Observers**, and press Return on your keyboard.


Raster Attribute Editor – intervis1_0A.img007284(:Layer_
File
Edit
Help
Layer Number:

Layer Number: 1

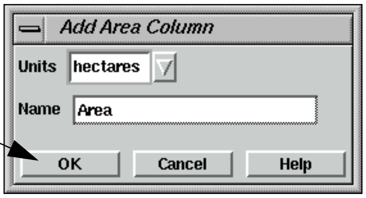
Row	Class_Names	Color	Red	Green
0			1	
1	Perimeter	🟥	1	
2	Hidden Region		1	
3	Visible by One Observer	🟦	0.639216	0.8470
4	Visible by Two Observers	🟦	0	

Class Names identify each of the areas in the viewshed

Add Area Column to the CellArray

1. Now select **Edit | Add Area Column** from the Raster Attribute Editor menu bar.

   The Add Area Column dialog opens.


Add Area Column
OK
Cancel
Help

Units: hectares

Name: Area

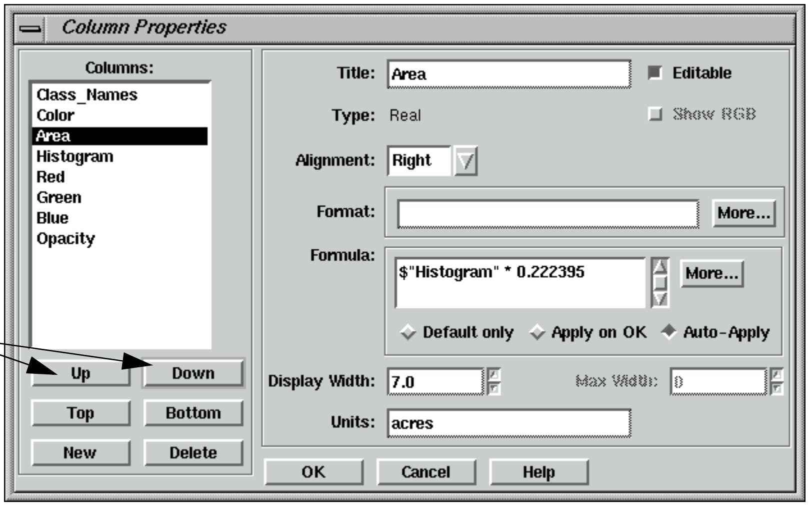
Click OK to accept hectares

2. Select **acres** from the **Units** popup list and click **OK** to dismiss the Add Area Column dialog.

   The **Area** column is added to end of the Raster Attributes CellArray.

3. Select **Edit | Column Properties** from the Raster Attribute Editor menu bar.

   The Column Properties dialog opens.


Column Properties
Columns:
Class_Names
Color
Area
Histogram
Red
Green
Blue
Opacity
Up
Down
Top
Bottom
New
Delete
OK
Cancel
Help
More...
More...

Title: Area ☑ Editable

Type: Real ☐ Show RGB

Alignment: Right

Format:

Formula: $"Histogram" * 0.222395

○ Default only ○ Apply on OK ● Auto-Apply

Display Width: 7.0 Max Width: 0

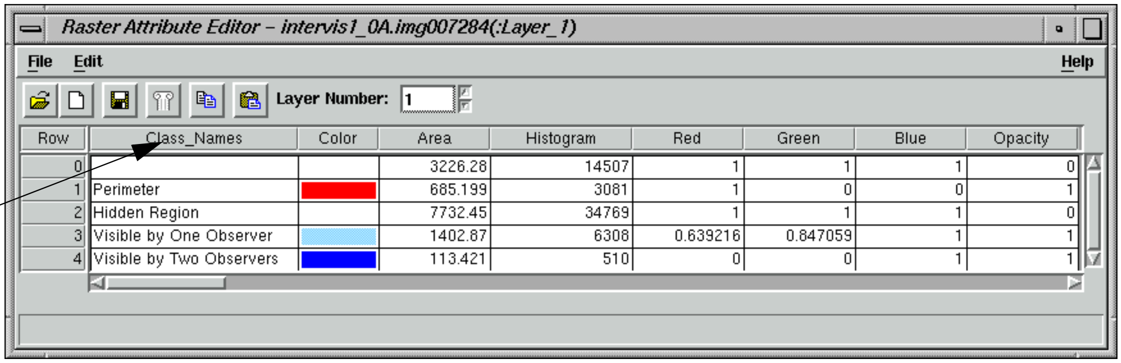
Units: acres

Use the Up and Down buttons to rearrange columns

4. In the **Columns** field, select **Histogram**, and click the **Up** button four times to move it below the **Color** column.

5. Now select **Area**, and click the **Up** button until it is between **Color** and **Histogram**.

6. Click **OK** to apply these changes to the CellArray. The Column Properties dialog closes.


Raster Attribute Editor – intervis1_0A.img007284(:Layer_1)
File
Edit
Help
Layer Number: 1

Row	Class_Names	Color	Area	Histogram	Red	Green	Blue	Opacity
0			3226.28	14507	1	1	1	0
1	Perimeter	🟥	685.199	3081	1	0	0	1
2	Hidden Region		7732.45	34769	1	1	1	0
3	Visible by One Observer	▨	1402.87	6308	0.639216	0.847059	1	1
4	Visible by Two Observers	🟦	113.421	510	0	0	1	1

Columns are displayed in the new order

7. Select **File | Save** in the Raster Attribute Editor to save all edits to the CellArray.

   You can now easily view the size and location of visible and hidden areas in the viewsheds.

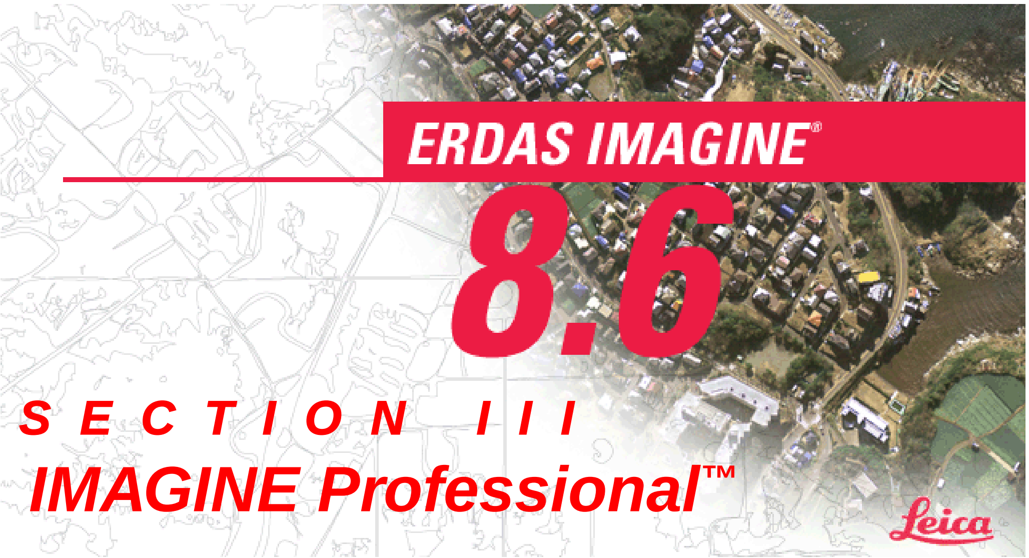
Query the Viewshed Layer

1. Click an area inside the Viewer. The corresponding class is highlighted in the CellArray of the Raster Attribute Editor dialog.

2. When you complete your query of the data, select **File | Close** in the Raster Attribute Editor to dismiss the dialog.

Finish

1. Click **Close** in the Viewshed dialog.
2. Select **File | Close** in the Viewer.
3. Select **File | Close Image Drape** in the Image Drape viewer.


ERDAS IMAGINE®
8.6
Leica

SECTION III

IMAGINE Professional™

CHAPTER SIXTEEN

Spatial Modeler

Introduction

In ERDAS IMAGINE, GIS analysis functions and algorithms are accessible through three main tools:

- script models created with the Spatial Modeler Language (SML)
- graphical models created with Model Maker
- pre-packaged functions in Image Interpreter

Spatial Modeler Language

SML is the basis for all GIS functions in ERDAS IMAGINE, and it is the most powerful. It is a modeling language that allows you to create script (text) models for a variety of applications. Using models, you can create custom algorithms that best suit your data and objectives.

Model Maker

Model Maker is essentially the SML with a graphical interface. This enables you to create graphical models using a palette of easy-to-use tools. Graphical models can be run, edited, saved, or converted to script form and edited further using the SML.

This tour guide focuses on Model Maker.

Image Interpreter

The Image Interpreter houses a set of common functions that are created using either Model Maker or the SML. They have been given a dialog interface to match the other processes in ERDAS IMAGINE. In most cases, you can run these processes from a single dialog. However, the actual models are also delivered with the software, so that you can edit them if you want more customized processing.

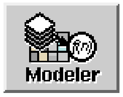
For more information on Image Interpreter functions, see "Chapter 11: Image Interpreter" *on page 237.*

Approximate completion time for this tour guide is 3 hours.

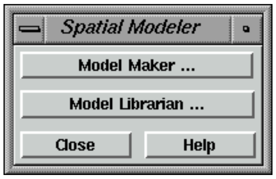
Start Model Maker

ERDAS IMAGINE should be running and a Viewer should be open.

1. Click the Modeler icon on the ERDAS IMAGINE icon panel.



The **Spatial Modeler** menu displays.



2. Click on **Model Maker** in the **Spatial Modeler** menu to start Model Maker.

The Model Maker viewer and tool palette open.

ERDAS IMAGINE is delivered with several sample graphical models that you can use as templates to create your own models. Open these models in Model Maker by selecting **File | Open** from the Model Maker viewer menu bar or clicking the Open icon on the tool bar.

3. Click **Close** in the **Spatial Modeler** menu to clear it from the screen.

Create Sensitivity Layer

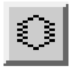
When three input thematic layers are combined, the resulting file has meaningful class values. These values may also be easily color coded in the final output file so that they are visible over the SPOT panchromatic reference data.

Therefore, you recode the data values of the input files so that the most environmentally sensitive areas have the highest class value and the least have the lowest value. You use class values 0-4, with 4 being the most environmentally sensitive and 0 being the least. This recode also facilitates defining the conditional statement within the function. These recodes are done at the same time the files are defined in the Raster dialog.

ou must have Model Maker running.

NOTE: Refer to the model pictured on page 341 when going through the following steps.

1. Click on the Raster icon in the Model Maker tool palette.



2. Click on the Lock icon.

3. Click in the Model Maker viewer in four different places to place three input Raster graphics and one output Raster graphic.

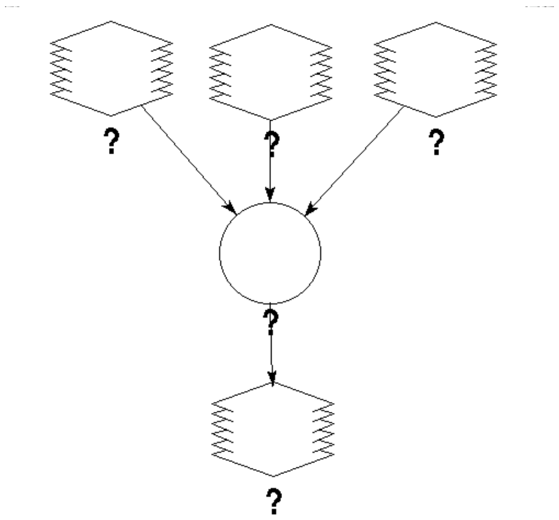
4. Select the Function icon in the Model Maker tool palette.

5. Click in the Model Maker viewer window to place a Function graphic on the page between the three inputs and the one output Raster graphic.

6. Select the Connect icon in the Model Maker tool palette.

7. Connect the three input Raster graphics to the Function and the Function to the output Raster by simply dragging from one graphic to another.

   Your model should look similar to the following example:

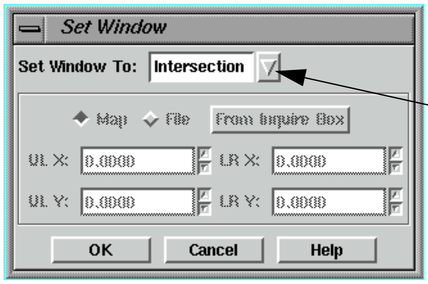


8. In the Model Maker tool palette, click the Lock icon to disable the lock tool.

9. Click the Select icon.

10. In the Model Maker viewer menu bar, select **Model | Set Window** to define the working window for the model.

    The Set Window dialog opens.


Set Window
Map
File
From Inquire Box
UL X:
0.0000
LR X:
0.0000
UL Y:
0.0000
LR Y:
0.0000
OK
Cancel
Help

Set Window To: Intersection

Click on this popup
list to select Intersection

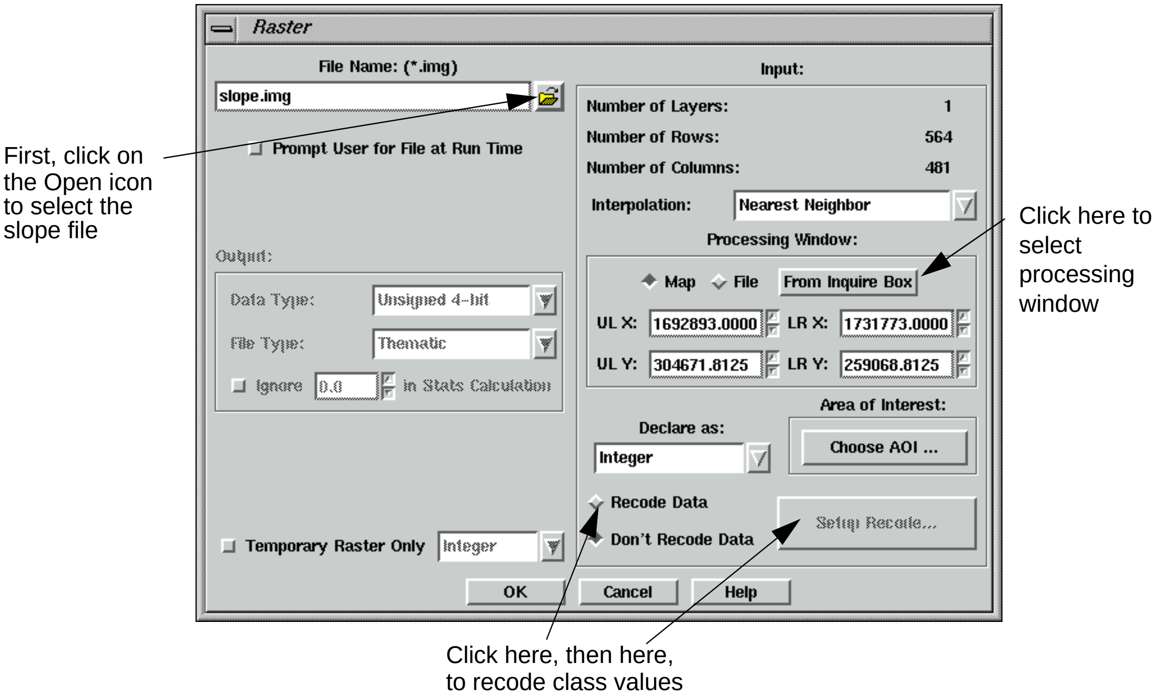
You want the model to work on the intersection of the input files. The default setting is the union of these files.

11. In the Set Window dialog, click on the **Set Window To** popup list and select **Intersection**.
12. Click **OK** in the Set Window dialog.

Define Input Slope Layer

1. In the Model Maker viewer, double-click the first input Raster graphic.

   The graphic is highlighted and the Raster dialog opens.



Raster

File Name: (*.img) slope.img

☐ Prompt User for File at Run Time

Output:

Data Type: Unsigned 4-bit

File Type: Thematic

☐ Ignore 0.0 in Stats Calculation

☐ Temporary Raster Only Integer

Input:

Number of Layers: 1

Number of Rows: 564

Number of Columns: 481

Interpolation: Nearest Neighbor

Processing Window:

● Map ○ File From Inquire Box

UL X: 1692893.0000 LR X: 1731773.0000

UL Y: 304671.8125 LR Y: 259068.8125

Area of Interest:

Declare as: Integer

○ Recode Data

● Don't Recode Data

First, click on the Open icon to select the slope file

Click here to select processing window

Click here, then here, to recode class values

2. In the Raster dialog, click on the Open icon under **File Name**.

The File Name dialog opens.

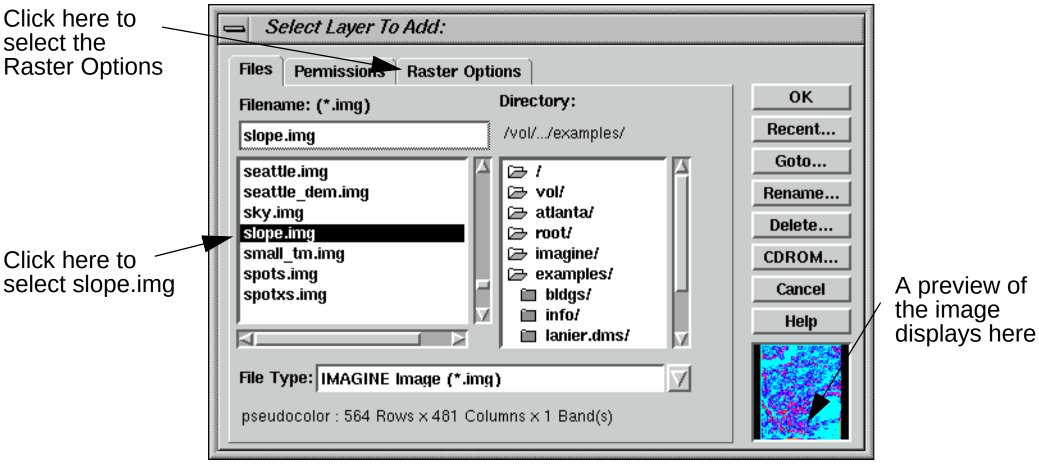
3. In the File Name dialog under **Filename**, click on the file **slope.img** and then click **OK**.

This image has some noise around the edges that you want to eliminate, so you use a subset of this image in the model. To take a subset, you display the file in a Viewer and select the processing window with an inquire box.

Display Slope Layer

1. Click the Open icon in a Viewer (or select **File | Open | Raster Layer** from the menu bar).

The Select Layer To Add dialog opens.


Click here to select the Raster Options
Click here to select slope.img
A preview of the image displays here
Select Layer To Add:
Files
Permissions
Raster Options
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
seattle.img
seattle_dem.img
sky.img
slope.img
small_tm.img
spots.img
spotxs.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/

Filename: (*.img) slope.img

Directory: /vol/.../examples/

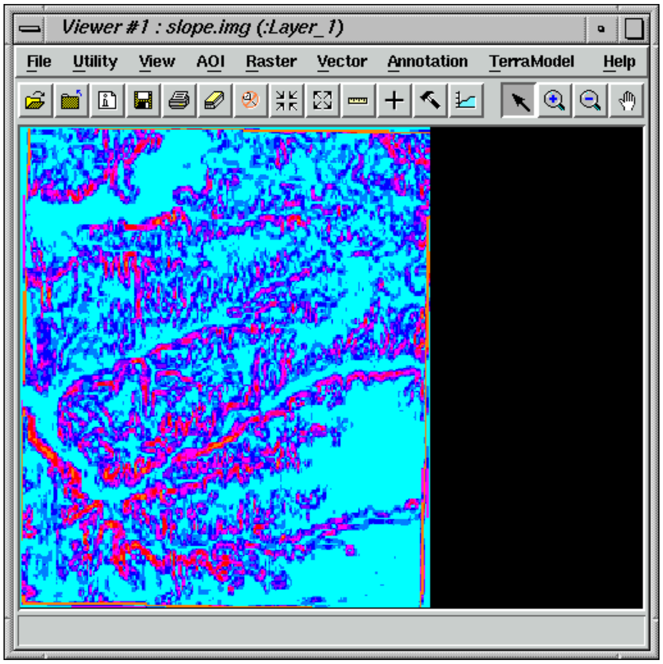
File Type: IMAGINE Image (*.img)

pseudocolor : 564 Rows x 481 Columns x 1 Band(s)

2. In the Select Layer To Add dialog under **Filename**, click on the file **slope.img**.

3. Click on the **Raster Options** tab at the top of the dialog, and then select the **Fit to Frame** option.

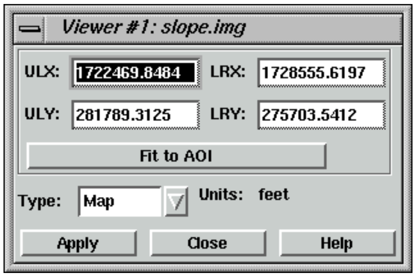
4. Click **OK** in the Select Layer To Add dialog to display the file in the Viewer.


Viewer #1 : slope.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Select Area to Use

1. With your cursor in the Viewer, right-hold **Quick View | Inquire Box**.

A white inquire box opens near the center of the image displayed in the Viewer. The Inquire Box Coordinates dialog also opens. The title of this dialog is Viewer #1: **slope.img**.



Viewer #1: slope.img

ULX: 1722469.8484 LRX: 1728555.6197

ULY: 281789.3125 LRY: 275703.5412

Type: Map Units: feet

2. Hold inside the inquire box in the Viewer and drag the box to the desired image area. You use the entire image area you select, except for the edges.

   You can reduce or enlarge the inquire box by dragging on the sides or corners.

   NOTE: You may wish to select nearly the entire image area with the inquire box, as this is helpful when you compare your output image with the example output image at the end of this exercise.

3. In the Raster dialog, under **Processing Window**, click **From Inquire Box**.

   The coordinates in the Raster dialog now match the coordinates in the Inquire Box Coordinates dialog.

4. Click **Close** in the Inquire Box Coordinates dialog.

Recode Classes

Now that the processing window is defined, you can recode the values.

1. In the Raster dialog, click the **Recode Data** option.

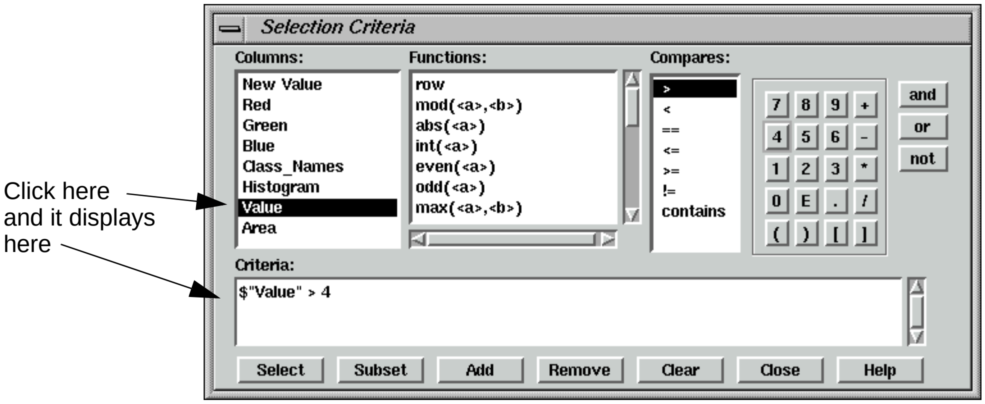
2. Click the **Setup Recode** button.

   The Recode dialog opens.

   You recode this file so that the classes with a slope greater than 25% have a class value of 1 and all other classes are 0 (zero). This is easy to do using the **Criteria** option of the **Row Selection** menu.

3. With your cursor in the **Value** column of the Recode dialog, right-hold **Row Selection | Criteria**.

   The Selection Criteria dialog opens.


Selection Criteria
Columns:
New Value
Red
Green
Blue
Class_Names
Histogram
Value
Area
Functions:
row
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
max(<a>,<b>)
Compares:
>
<
==
<=
>=
!=
contains
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
and
or
not
Criteria:
Select
Subset
Add
Remove
Clear
Close
Help

Criteria: $"Value" > 4

Click here and it displays here

Next, you select all classes with a slope greater than 25%. By looking at the Recode dialog, you can see that all classes greater than **Value 4** have a slope greater than 25%. You can then invert your selection to recode all classes with values less than 25%.

4. In the Selection Criteria dialog, under **Columns**, click **Value**.

   $ “Value” is displayed in the **Criteria** window at the bottom of the dialog.

5. Under **Compares**, click **>**.

6. In the calculator, click the number **4**.

   The **Criteria** window now shows **$ “Value” > 4**.

7. In the Selection Criteria dialog, click **Select** to select all classes meeting that criteria in the Recode dialog.

   All classes greater than **4** are highlighted in yellow in the Recode dialog.

8. Click **Close** in the Selection Criteria dialog.

9. In the Recode dialog, confirm that the **New Value** is set to **1**.

10. In the Recode dialog, click **Change Selected Rows** to give the selected classes a new value of **1**.

11. With your cursor in the **Value** column of the Recode dialog, right-hold **Row Selection | Invert Selection** to deselect all currently selected classes and select all nonselected classes.

12. Enter a **New Value** of **0** in the Recode dialog.

13. Click **Change Selected Rows** to give the selected classes a new value of **0**.

14. Click **OK** in the Recode dialog.

    The Recode dialog closes.

15. Click **OK** in the Raster dialog.

The Raster dialog closes. The Raster graphic in the Model Maker viewer now has **n1_slope_RC** written under it.

Define Input Flood Plain Layer

1. Double-click the second Raster graphic in the Model Maker viewer.

   The graphic is highlighted and the Raster dialog opens.

2. In the Raster dialog, click on the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, select the file **floodplain.img** and then click **OK**.

   This file does not need to be subset or recoded.

4. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n2_floodplain** is written underneath the second Raster graphic.

Define Input Land Cover Layer

1. Double-click the third Raster graphic in the Model Maker viewer.

   The graphic is highlighted and the Raster dialog opens.

2. In the Raster dialog, click on the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, select the file **landcover.img** and then click **OK**.

   You recode this file so that the most sensitive areas have the highest class value.

4. In the Raster dialog, click the **Recode Data** option.
5. Click the **Setup Recode** button.

   The Recode dialog opens.

6. In the **Value** column of the Recode dialog, click on **1** to select it.
7. In the **New Value** box, enter a **New Value** of **4**.

8. Click **Change Selected Rows** to recode **Riparian** to **4**.

   Now both **Riparian** and **Wetlands** have a class value of **4**.

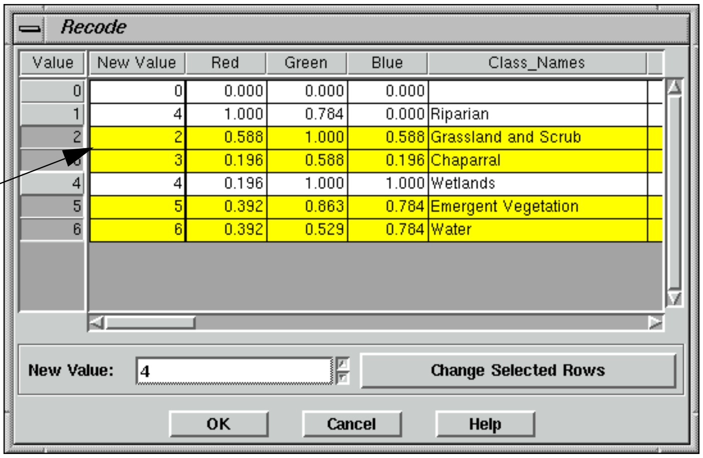
9. With your cursor in the **Value** column, right-hold **Row Selection | Invert Selection**.

   Now all classes are selected except one (**Riparian**).

10. With your cursor in the **Value** column, Shift-click on **4** to deselect **Wetlands**.

11. With your cursor in the **Value** column, Shift-click on **0** to deselect the background.

    Your Recode dialog looks like the following:



Value	New Value	Red	Green	Blue	Class_Names
0	0	0.000	0.000	0.000	
1	4	1.000	0.784	0.000	Riparian
2	2	0.588	1.000	0.588	Grassland and Scrub
3	3	0.196	0.588	0.196	Chaparral
4	4	0.196	1.000	1.000	Wetlands
5	5	0.392	0.863	0.784	Emergent Vegetation
6	6	0.392	0.529	0.784	Water

New Value: 4

Rows in yellow are recoded to a value of 1

12. Enter a **New Value** of **1**.

13. Click **Change Selected Rows**.

14. Click **OK** to close the Recode dialog.

15. In the Raster dialog, click **OK**.

    n3_landcover_RC is written under the third Raster graphic in the Model Maker viewer.

    Now, all of the files are set up so that the most sensitive areas have the higher class values:

Table 16-1: Class Values for n3_landcover_RC

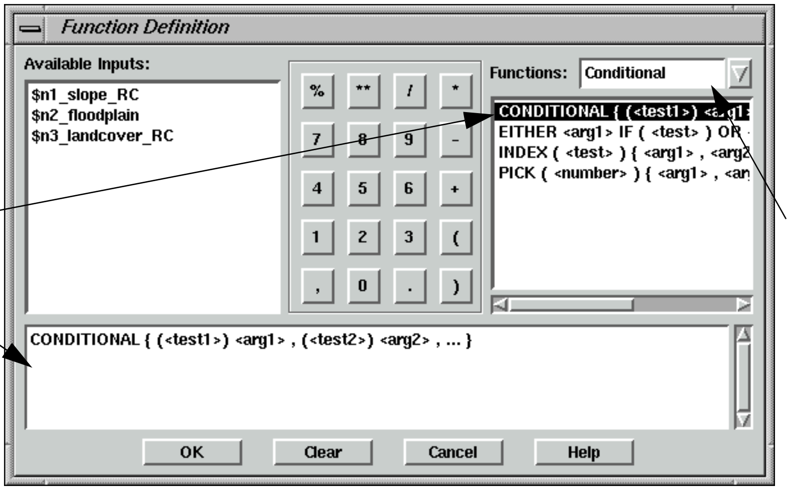
Class	Value
> 25 percent slope	1
flood plain	1
riparian & wetlands	4
undeveloped land	1

These values are used in the next step to create the sensitivity file.

Define Function

1. In the Model Maker viewer, double-click the Function graphic.

   The graphic is highlighted and the Function Definition dialog opens.


Function Definition
Available Inputs:
$n1_slope_RC
$n2_floodplain
$n3_landcover_RC
% ** / *
7 8 9 -
4 5 6 +
1 2 3 (
, 0 .)
Functions:
Conditional
CONDITIONAL { (<test1>) <arg1
EITHER <arg1> IF (<test>) OR
INDEX (<test>) { <arg1> , <arg2
PICK (<number>) { <arg1> , <ar
OK
Clear
Cancel
Help

CONDITIONAL { (<test1>) <arg1> , (<test2>) <arg2> , ... }

Click here and it displays here

Select the Conditional Functions here

Next, you use a conditional statement to create a new file that contains only the environmentally sensitive areas.

2. In the Function Definition dialog, click on the **Functions** popup list and select **Conditional**.

3. Click on **CONDITIONAL** in the box below **Functions**.

   The **CONDITIONAL** function is placed in the function definition window at the bottom of the dialog.

4. Type the following statement in the definition box, replacing the previously created condition statement:

```
CONDITIONAL { ($n3_landcover_RC==0)0,
                        ($n3_landcover_RC==4)4,
                        ($n1_slope_RC==1)3,
                        ($n2_floodplain==1)2,
                        ($n3_landcover_RC==1)1 }
```

   NOTE: The file names can be added to your function definition simply by clicking in the appropriate spot in the function definition, and then clicking on the file name in the list of **Available Inputs**.

This creates a new output file with the class values 0-4. Each class contains the following:

Table 16-2: Conditional Statement Class Values

Class	Contents
0	developed
1	undeveloped land
2	flood plain
3	> 25 percent slope
4	riparian & wetlands

Areas with a class value of 4 are the most environmentally sensitive, and are therefore unsuitable for development. Classes 3-1 are also environmentally sensitive, but proportionally less so. Further analysis determines whether classes 3-1 are eligible for development.

5. Take a moment to check over the conditional statement you just entered to be sure it is 100% accurate. The model does not run if the information has not been entered accurately.

6. Click **OK** in the Function Definition dialog.

   The Function Definition dialog closes and **CONDITIONAL** is written under the Function graphic.

Define Output Raster Layer

1. In the Model Maker viewer, double-click the output Raster graphic.

   The graphic is highlighted and the Raster dialog opens.

2. Under **File Name**, type the name **sensitivity.img** for the new output file.

   NOTE: Be sure that you specify a directory in which you have write permission.

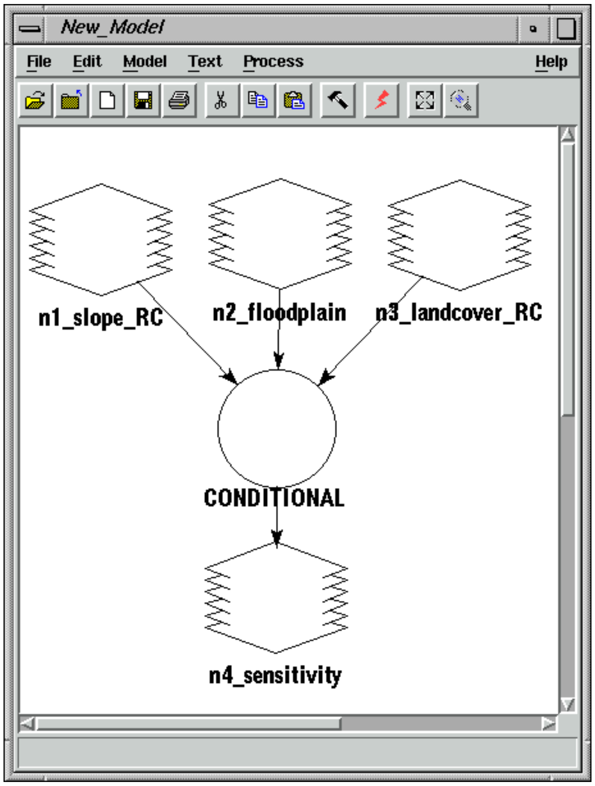
3. Click the **Delete if Exists** option so that the output file is automatically overwritten when the model is run again.

4. Click on the **File Type** popup list and select **Thematic**.

5. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n4_sensitivity** is written under the output Raster graphic in the Model Maker viewer.

   Your model should look similar to the following example:

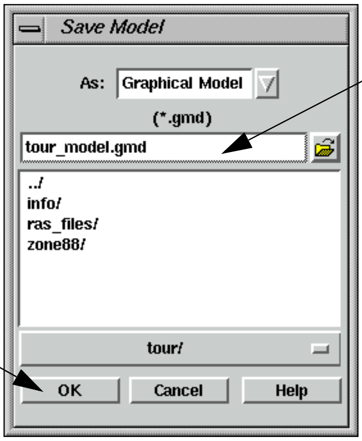

New_Model
File
Edit
Model
Text
Process
Help
n1_slope_RC
n2_floodplain
n3_landcover_RC
CONDITIONAL
n4_sensitivity

Save and Run the Model

1. In the Model Maker viewer tool bar, click the Save icon (or select **File | Save As** from the Model Maker viewer menu bar) to save the model.

   The Save Model dialog opens.

2. Enter a name for the model. Be sure you are saving in a directory in which you have write permission.



Enter the name of the model here

Click OK

3. Click **OK** in the Save Model dialog.

Run the Model

You can now run this portion of the model to see if it works correctly.

1. In the Model Maker viewer tool bar, click the Run icon (or select **Process | Run** from the Model Maker viewer menu bar) to run the model.

While the model runs, a Job Status dialog opens, reporting the status of the model.

2. When the model is finished, click **OK** in the Job Status dialog.

Enhance SPOT Data

To enhance the detail in the SPOT data, you run a convolution kernel over it before it is combined with the sensitivity layer. This portion of the model includes a Raster input, a Matrix input, a Function, and a Raster output.

Follow the next series of steps to create this portion of the model in a new Model Maker viewer. After you have verified that this portion runs correctly, you paste it into the first Model Maker viewer.

NOTE: Refer to the model pictured on page 353 when going through the following steps.

1. Click the new Window icon in the Model Maker viewer tool bar or select **File | New** to create a new Model Maker viewer.

The new Model Maker viewer opens.

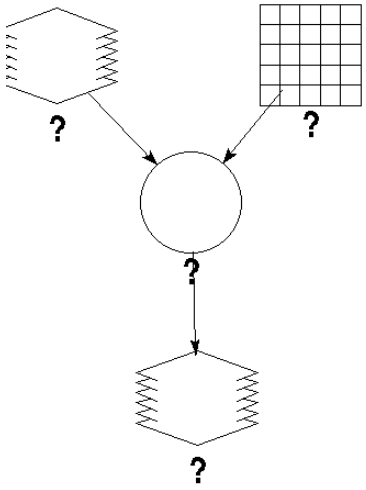
2. Click on the Raster icon in the Model Maker tool palette, then click the Lock icon.

3. Click twice in the Model Maker viewer to place the input and output Raster graphics.
4. Click on the Matrix icon in the Model Maker tool palette.
5. Click in the Model Maker viewer to place the input Matrix graphic.

   This is where you define the convolution kernel.
6. Click on the Function icon in the Model Maker tool palette.
7. Click on the Model Maker viewer to place a Function graphic on the page.

   Place the Function graphic between the two inputs and the output Raster graphic.
8. Click on the Connect icon.
9. Connect the input Raster graphic to the Function, the input Matrix to the Function, and the Function to the output Raster.

   This part of the model looks similar to the following example:



10. In the Model Maker tool palette, click the Lock icon to disable the Lock tool.
11. Click the Select icon.

Define Input SPOT Layer

1. Double-click the input Raster graphic in the Model Maker viewer.

   The graphic is highlighted and the Raster dialog opens.

2. In the Raster dialog, click on the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, click on the file **spots.img** and then click **OK**.
4. Click **OK** in the Raster dialog.

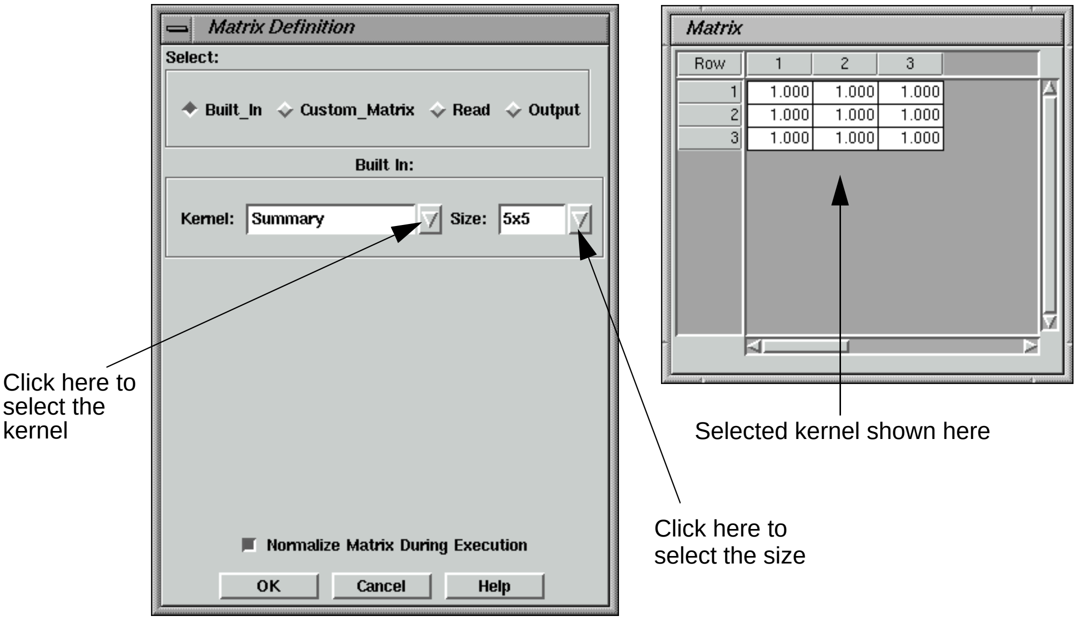
   The Raster dialog closes, and **n1_spots** is written under the input Raster graphic.

Define Input Convolution Kernel

In Model Maker, you have access to built-in kernels or you can create your own. In this exercise, use the built-in 5 × 5 summary filter.

1. Double-click the input Matrix graphic.

   The Matrix Definition and Matrix dialogs open.


Matrix Definition
Select:
◆ Built_In ◇ Custom_Matrix ◇ Read ◇ Output
Built In:
OK
Cancel
Help
Matrix

Kernel: Summary

Size: 5x5

☒ Normalize Matrix During Execution

Row	1	2	3
1	1.000	1.000	1.000
2	1.000	1.000	1.000
3	1.000	1.000	1.000

Click here to select the kernel

Selected kernel shown here

Click here to select the size

2. In the Matrix Definition dialog, click on the **Kernel** popup list and select **Summary**.
3. Click on the **Size** popup list and select **5x5**.

   The kernel displays in the Matrix dialog.
4. Click **OK** in the Matrix Definition dialog.

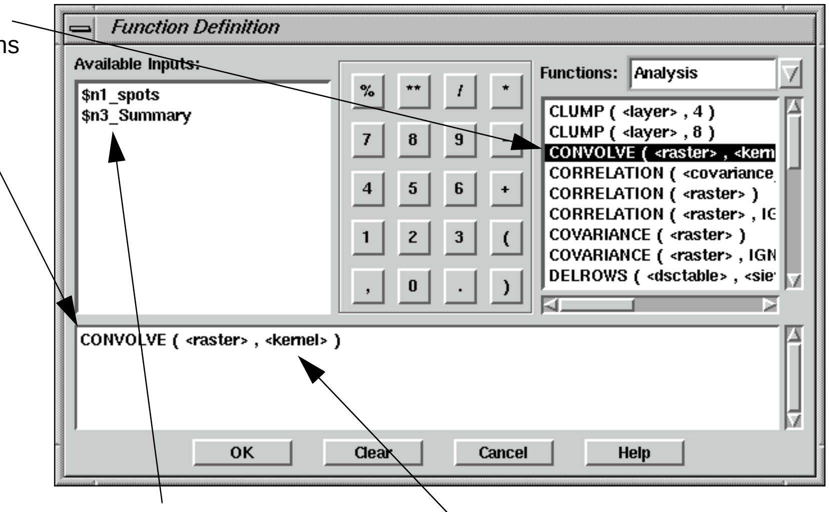
   The Matrix Definition and Matrix dialogs close, and **n3_Summary** is written under the Matrix graphic in the Model Maker viewer.

Define Function

1. Double-click the Function graphic in the Model Maker viewer.

   The Function Definition dialog opens.

Click here and it opens here


Function Definition
Available Inputs:
$n1_spots
$n3_Summary
% ** / *
7 8 9 -
4 5 6 +
1 2 3 (
, 0 .)
Functions:
Analysis
CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie
OK
Clear
Cancel
Help

CONVOLVE (<raster> , <kernel>)

Click here and inputs automatically display in the designated prototype here

2. Click **CONVOLVE** from the list below **Functions**.

   The **CONVOLVE** statement displays in the function definition window.

3. Click in the first prototype (**<raster>**), and then click on **$n1_spots** under **Available Inputs** to define the raster input.

4. Click in the second prototype (**<kernel>**), and then click on **$n3_Summary** under **Available Inputs** to define the kernel.

5. Click **OK** to close the Function Definition dialog.

   The Function Definition dialog closes and **CONVOLVE** is written below the Function graphic in the Model Maker viewer.

Define Output Raster Layer

1. Double-click the output Raster graphic in the Model Maker viewer.

   The Raster dialog opens.

2. In the Raster dialog under **File Name**, type the name **spot_summary** for the new output file. The .img extension is added automatically.

   Be sure that you specify a directory in which you have write permission.

3. Click on the **Delete if Exists** option.

4. Confirm that **Continuous** is selected for the **File Type**.

5. Click **OK** in the Raster dialog.

The Raster dialog closes and **n2_spot_summary** is written under the Raster graphic in the Model Maker viewer.

Save and Run the Model

1. In the Model Maker viewer tool bar, click the Save icon (or select **File | Save As** from the Model Maker viewer menu bar) to save the model.

   The Save Model dialog opens.

2. Enter a name for the model, such as **convolve.gmd**, being sure that you specify a directory in which you have write permission.

3. Click **OK** in the Save Model dialog.

Run the Model

You can now run this portion of the model to see if it works correctly.

1. In the Model Maker viewer tool bar, click the Run icon (or select **Process | Run** from the Model Maker viewer menu bar) to run the model.

   While the model runs, a Status box opens, reporting the status of the model.

2. When the model is finished running, click **OK** in the Status box.

Combine Models

You now use the Copy and Paste commands to combine these two separate models into one. Make sure that both models you created are open.

1. In the menu bar of the second model you created, select **Edit | Select All**.

   You can also select objects by clicking and dragging in the Model Maker viewer. All objects contained within the selection box that you draw are selected.

2. Click the Copy icon in the tool bar of the same model (or select **Edit | Copy** from the menu bar) to copy the selected objects to the paste buffer.

3. Click the Paste icon in the tool bar of the first model (or select **Edit | Paste** from the menu bar) to paste the second model into the first Model Maker viewer.

   The second model is pasted on top of the first model.

4. Close the second Model Maker viewer by selecting **File | Close**.

   NOTE: Do not select **File | Close All***, as this closes both of the models.*

5. Drag the pasted model to the right in the Model Maker viewer, so that it does not overlap the first model.

   You can resize the Model Maker viewer to see the entire model.

6. Click outside of the selection to deselect everything.

Combine Sensitivity Layer with SPOT Data

With both the thematic sensitivity layer (**sensitivity.img**) and the SPOT data (**spot_summary.img**) defined, you can use these two files as the input raster layers in a function that combines the two files into one final output. A Scalar is also used in the function to offset the data file values in the SPOT image by five, so that the sensitivity analysis does not overwrite any SPOT data.

NOTE: Refer to the model pictured on page 362 when going through the next set of steps.

1. Click on the Function icon in the Model Maker tool palette.

2. Click in the Model Maker viewer below the output raster graphics (**n4_sensitivity** and **n7_spot_summary**) to place a Function graphic.

3. Click on the Scalar icon in the Model Maker tool palette.

4. Click in the Model Maker viewer to the left of the Function graphic you just positioned to place an input Scalar.

5. Click on the Raster icon in the Model Maker tool palette.

6. Click in the Model Maker viewer below the Function to place an output Raster graphic.

7. Click on the Connect icon and then on the Lock icon.

8. Connect the input Raster graphics (**n4_sensitivity** and **n7_spot_summary**) to the Function, the input Scalar to the Function, and then the Function to the output Raster.

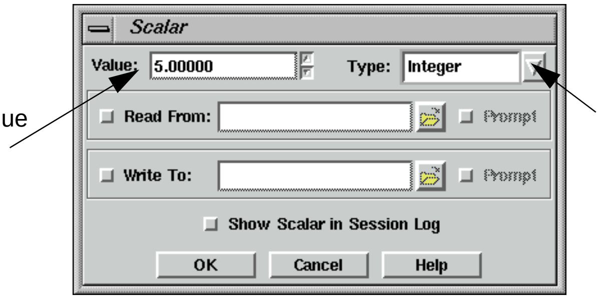
9. Click the Lock icon to disable the lock tool.

10. Click the Select icon.

Define Input Scalar

1. Double-click the Scalar graphic in the Model Maker viewer.

   The Scalar dialog opens.



Scalar

Value: 5.00000

Type: Integer

☐ Read From: ☐ Prompt

☐ Write To: ☐ Prompt

☐ Show Scalar in Session Log

Enter value here

Change scalar type here

2. In the Scalar dialog, enter a **Value** of **5**.

3. Click on the **Type** popup list and select **Integer**.

4. Click **OK** in the Scalar dialog.

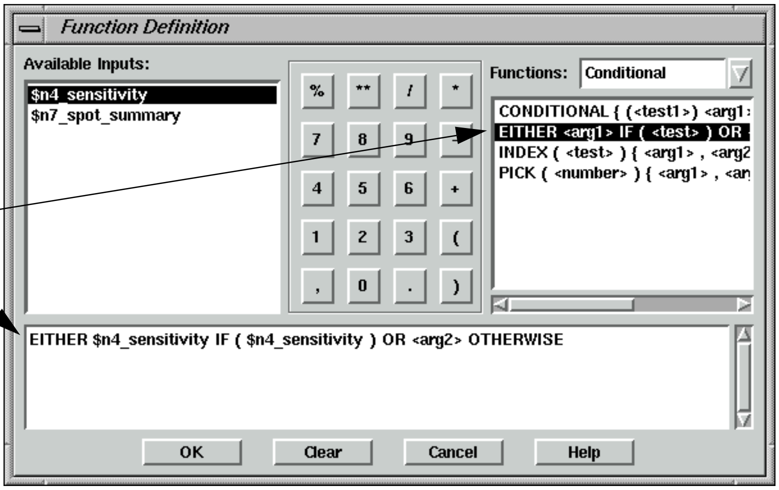
   The Scalar dialog closes and **n11_Integer** displays under the Scalar graphic in the Model Maker viewer.

Define Function

Next, you create a file that shows sensitivity data where they exist and allows the SPOT data to show in all other areas. Therefore, you use the conditional statement.

1. Double-click the untitled Function graphic in the Model Maker viewer.

   The Function Definition dialog opens.


Function Definition
Available Inputs:
$n4_sensitivity
$n7_spot_summary
%
**
/
*
7
8
9
-
4
5
6
+
1
2
3
(
,
0
.
)
Functions:
Conditional
CONDITIONAL { (<test1>) <arg1>
EITHER <arg1> IF (<test>) OR
INDEX (<test>) { <arg1> , <arg2
PICK (<number>) { <arg1> , <ar
OK
Clear
Cancel
Help

EITHER $n4_sensitivity IF ($n4_sensitivity) OR <arg2> OTHERWISE

When you click here,
it displays here

2. In the Function Definition dialog, click on the **Functions** popup list and select **Conditional**.

3. In the list under **Functions**, click on **EITHER**.

   The **EITHER** statement and prototype arguments display in the function definition window.

4. Click in the first prototype **<arg1>**, then click **$n4_sensitivity** under **Available Inputs** to automatically replace the prototype with an argument.

5. Click in the prototype **<test>**, then click **$n4_sensitivity**.

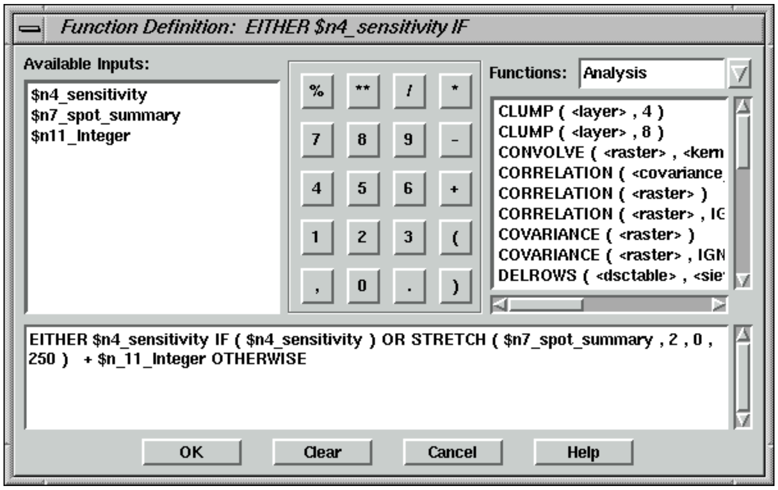
   The function definition now reads:

   EITHER $n4_sensitivity IF ($n4_sensitivity) OR <arg2> OTHERWISE

6. Click on the **Functions** popup list and select **Analysis**.

7. Click the remaining prototype, **<arg2>**, and then scroll down the list under **Functions** and click on the first **STRETCH** function to replace **<arg2>**.

   The **STRETCH** function and its prototype arguments are inserted into the function definition.


Function Definition: EITHER $n4_sensitivity IF
Available Inputs:
$n4_sensitivity
$n7_spot_summary
$n11_Integer
% ** / *
7 8 9 -
4 5 6 +
1 2 3 (
, 0 .)
Functions: Analysis
CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie
OK
Clear
Cancel
Help

EITHER $n4_sensitivity IF ($n4_sensitivity) OR STRETCH ($n7_spot_summary , 2 , 0 , 250) + $n_11_Integer OTHERWISE

8. Click on **<raster>**, then click on the file name **$n7_spot_summary** under **Available Inputs**.

9. Click on **<stdcount>**, then click on the number **2** on the calculator.

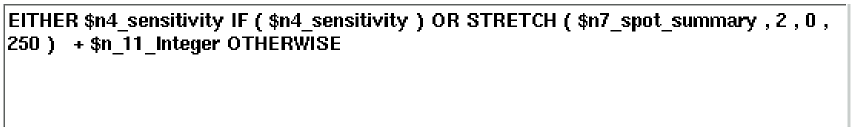
10. Using this same method, replace **<min>** with **0** and **<max>** with **250**.

The **STRETCH** function uses two standard deviations to stretch the data file values of **spot_summary.img** between 0 and 250. The scalar is added to ensure that there are no data file values between 0 and 4, since these are the values in the sensitivity file.

11. Click in front of **OTHERWISE** to insert the cursor in the function definition.

12. Click **+** on the calculator, then **$n11_Integer** under **Available Inputs**, to add the scalar to the function.

The final function definition should look like the following:



EITHER $n4_sensitivity IF ($n4_sensitivity) OR STRETCH ($n7_spot_summary , 2 , 0 , 250) + $n_11_Integer OTHERWISE

13. Click **OK** in the Function Definition dialog.

The Function Definition dialog closes, and **EITHER $n4_sensitivity IF** is written under the Function graphic in the Model Maker viewer.

Define Output Raster Layer

1. Double-click the untitled output Raster graphic.

The Raster dialog opens.

2. In the Raster dialog, enter the file name **sensitivity_spot** for the new output file.

   Be sure that you specify a directory in which you have write permission.

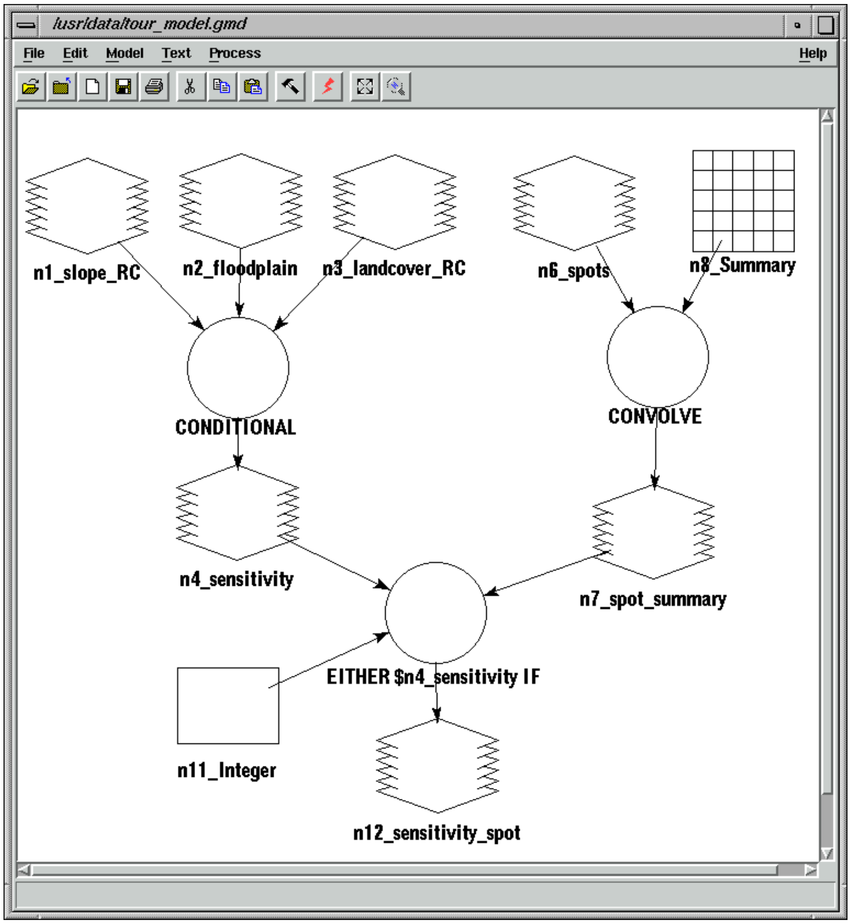
3. Click on the **Delete if Exists** option.

4. Click on the **File Type** popup list and select **Thematic**.

5. Click **OK** in the Raster dialog.

   The Raster dialog closes, and **n12_sensitivity_spot** is written under the Raster graphic in the Model Maker viewer.

   Your final model should look like the following example:


/usr/data/tour_model.gmd
File
Edit
Model
Text
Process
Help
n1_slope_RC
n2_floodplain
n3_landcover_RC
n6_spots
n8_Summary
CONDITIONAL
CONVOLVE
n4_sensitivity
n7_spot_summary
EITHER $n4_sensitivity IF
n11_Integer
n12_sensitivity_spot

Save and Run the Model

1. In the Model Maker viewer tool bar, click the Save icon (or select **File | Save** from the Model Maker viewer menu bar) to save the model.

Run the Model

You can now run the entire model.

2. In the Model Maker viewer tool bar, click the Run icon (or select **Process | Run** from the Model Maker viewer menu bar) to run the model.

While the model runs, a Job Status dialog opens, reporting the status of the model.

3. When the model is finished running, click **OK** in the Job Status dialog.

Display New Layer

Once your model has run, the new output file is created. You can display this file in a Viewer and modify the class colors and class names of the overlaid sensitivity analysis.

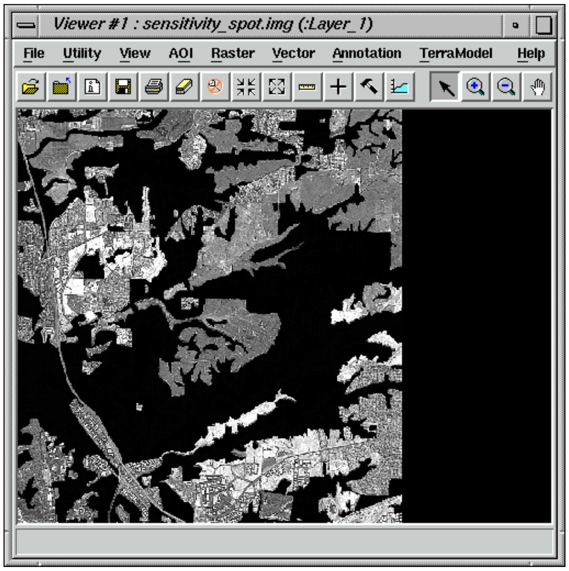
Prepare

You must have run the model and you must have a Viewer open.

1. In the Viewer tool bar, click the Open icon (or select **File | Open | Raster Layer** from the Viewer menu bar).

The Select Layer To Add dialog opens.

2. Under **Filename**, click the file **sensitivity_spot.img**.
3. Click on the **Raster Options** tab at the top of the dialog and confirm that the **Fit to Frame** option is selected, so that you can see the entire layer.
4. Click **OK** to display the file.

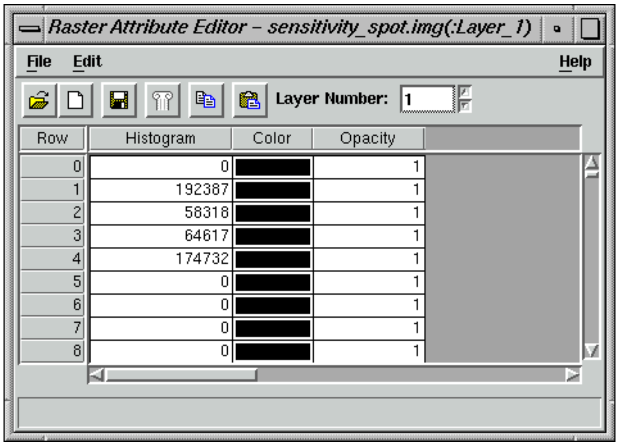

Viewer #1 : sensitivity_spot.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Adjust Colors

The sensitivity analysis is displayed with a gray scale color scheme.

1. In the Viewer menu bar, select **Raster | Attributes**.

   The Raster Attribute Editor opens. You add a Class Names column.


Raster Attribute Editor – sensitivity_spot.img(:Layer_1)
File
Edit
Help
Layer Number:
1

Row	Histogram	Color	Opacity
0	0	⬛	1
1	192387	⬛	1
2	58318	⬛	1
3	64617	⬛	1
4	174732	⬛	1
5	0	⬛	1
6	0	⬛	1
7	0	⬛	1
8	0	⬛	1

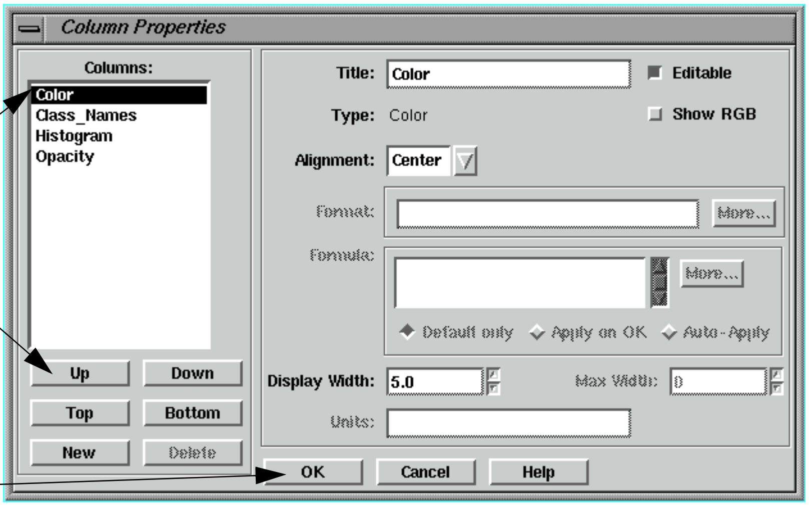
2. In the Raster Attribute Editor, select **Edit | Add Class Names**.

   A new **Class_Names** column is added to the CellArray.

   Next, rearrange the columns so that the **Color** and **Class Name** columns come first. This makes it easier to change the colors of the overlaid sensitivity analysis.

3. In the Raster Attribute Editor, select **Edit | Column Properties**.

The Column Properties dialog opens.



First, click on the column name here, then click on the action here

Click here to change the order of the columns

Columns:
- Color
- Class_Names
- Histogram
- Opacity

Title: Color

☑ Editable

Type: Color

☐ Show RGB

Alignment: Center

Format:

Formula:

◆ Default only ◇ Apply on OK ◇ Auto-Apply

Display Width: 5.0

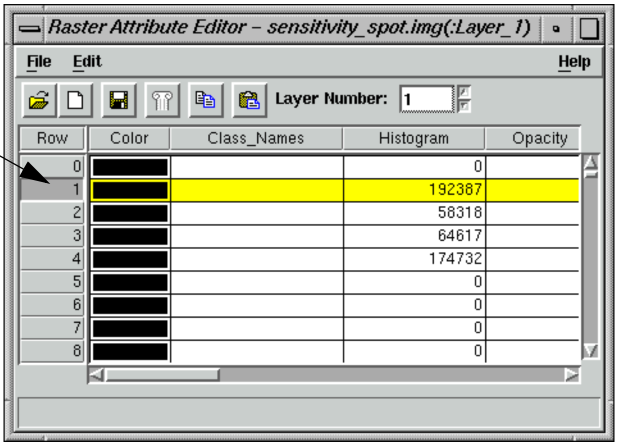
Max Width: 0

Units:

4. Click on **Color** under **Columns**, then click **Top** to make **Color** the first column in the Raster Attribute Editor.

5. Click **OK** in the Column Properties dialog to change the order of the columns.

   The Raster Attribute Editor now looks similar to the following example:



This is class 1, notice the Histogram value

Raster Attribute Editor - sensitivity_spot.img(:Layer_1)

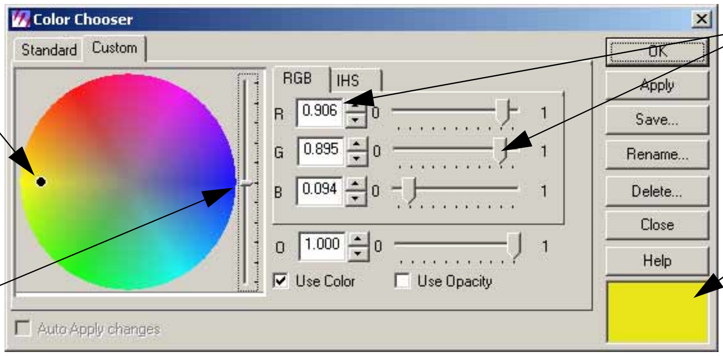
Layer Number: 1

Row	Color	Class_Names	Histogram	Opacity
0	⬛		0	
1	⬛		192387	
2	⬛		58318	
3	⬛		64617	
4	⬛		174732	
5	⬛		0	
6	⬛		0	
7	⬛		0	
8	⬛		0	

Next, change the colors and class names.

6. To change the color of the class 1, with your pointer over the color patch for that class, right-hold **Other**.

   The Color Chooser dialog opens.


Color Chooser
Standard
Custom
RGB
IHS
OK
Apply
Save...
Rename...
Delete...
Close
Help

R 0.906 0 1

G 0.895 0 1

B 0.094 0 1

O 1.000 0 1

☑ Use Color ☐ Use Opacity

☐ Auto Apply changes

Drag this dot to select a color on the color wheel

Drag this slider bar upward to enhance the RGB

Enter RGB values here or move the slider bars

A preview of the selected color displays here

This dialog gives you several options for changing the class colors. You can move the black dot on the color wheel, use the slider bars, select colors from a library (under the **Standard** tab), or enter RGB values.

7. Experiment with each of these methods to alter the class colors of classes 1 through 4. Change class 1 to **Green**, class 2 to **Yellow**, class 3 to **Tan**, and class 4 to **Red**.

When you have selected the desired color for a class, click **Apply** and then **Close** in the Color Selector dialog. Then redisplay the Color Chooser for the next class by moving your cursor to that color patch and right-holding a specific color or **Other**.

8. Click in the **Class_Names** column of class 1.

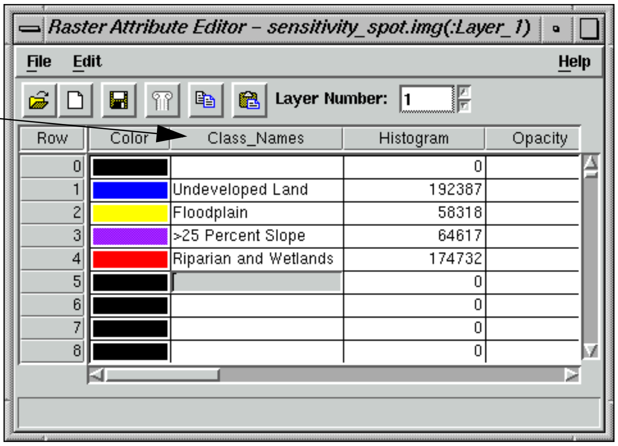
9. Type **Undeveloped Land**. Press Return on your keyboard.

Your cursor is now in the class name field of class 2.

10. Type **Floodplain** for class 2. Press Return.

11. Type **>25 Percent Slope** for class 3. Press Return.

12. Type **Riparian and Wetlands** for class 4. Press Return.


Raster Attribute Editor – sensitivity_spot.img(:Layer_1)
File
Edit
Help
Layer Number: 1

Row	Color	Class_Names	Histogram	Opacity
0	⬛		0	
1	🟦	Undeveloped Land	192387	
2	🟨	Floodplain	58318	
3	🟪	>25 Percent Slope	64617	
4	🟥	Riparian and Wetlands	174732	
5	⬛		0	
6	⬛		0	
7	⬛		0	
8	⬛		0	

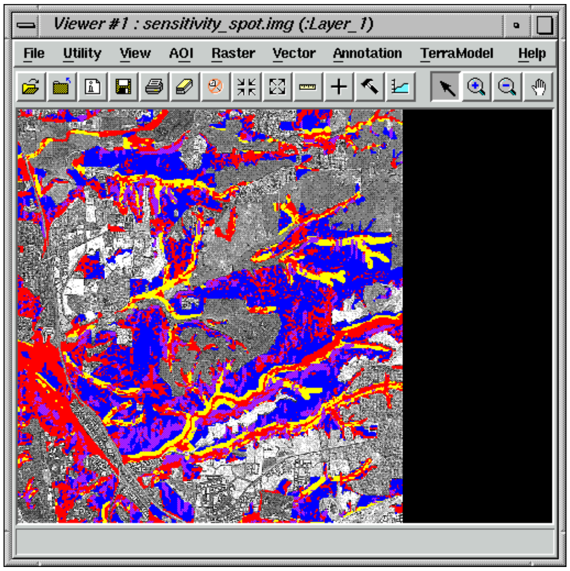
New class names go in this column

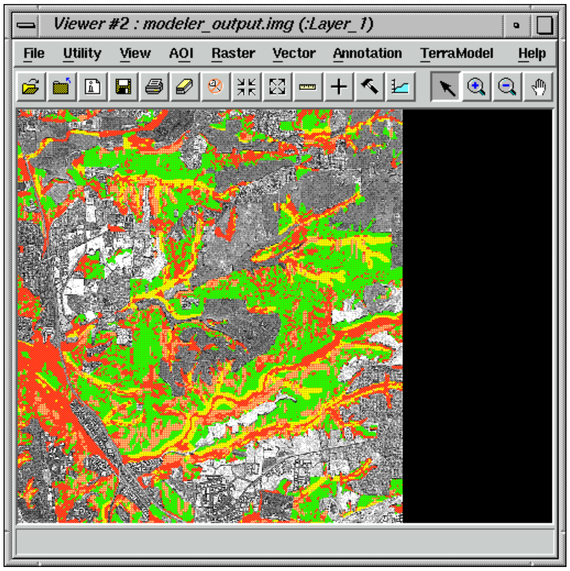
Test the Output

The following steps describe how to compare your output with the one delivered with ERDAS IMAGINE.

You must have completed the Spatial Modeler tour guide up to this point, creating **sensitivity_spot.img** in the process. The file **sensitivity_spot.img** should be displayed in a Viewer.

1. Display the file **<IMAGINE_HOME>/examples/modeler_output.img** in a second Viewer.
2. Select **Session | Tile Viewers** from the ERDAS IMAGINE menu bar to position the two Viewers side by side, so that you can view both images at once.


Viewer #1 : sensitivity_spot.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help


Viewer #2 : modeler_output.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

3. In Viewer #1, select **View | Link/Unlink Viewers | Geographical**.

   A Link/Unlink Instructions dialog opens, instructing you to click in Viewer #2 to link the two Viewers.

4. Click in Viewer #2 to link the two Viewers and close the Link/Unlink Instructions dialog.

   If **sensitivity_spot.img** *is a subset of* **modeler_output.img**, *a white bounding box displays in Viewer #2 (***modeler_output.img***), marking the area of the image that is shown in Viewer #1 (***sensitivity_spot.img***).*

5. Select **Utility | Inquire Cursor** from either Viewer's menu bar.
6. Compare the two images using the Inquire Cursor.
7. When you are finished, click **Close** in the Inquire Cursor dialog.
8. Right-click in the Viewer displaying **sensitivity_spot.img** to access the **Quick View** menu.

9. Select **Geo Link/Unlink**.

10. Click in the Viewer containing **modeler_output.img** to break the link.

Add Annotation to a Model

You can add annotation to a model to make it more understandable to others, or to help you remember what the model does. It is also a helpful organizational tool if you create several models and need to keep track of them all.

Next, add a title and an explanation of each function to the model you just created.

You must have the model open.

NOTE: Refer to the model pictured on page 370 when going through the next set of steps.

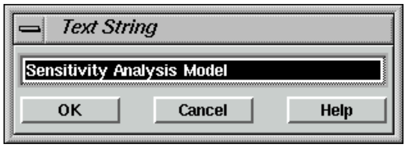
Add a Title

1. Select the Text icon in the Model Maker tool palette.

A

2. Click near the center of the top of the model page to indicate where you want to place the text.

   The Text String dialog opens.



Sensitivity Analysis Model

3. Type these words in the Text String dialog: Sensitivity Analysis Model

4. Press Return on your keyboard, and then click **OK** in the Text String dialog.

   The text string you typed in step 3. displays on the page.

Format Text

1. Click on the text string you just added to select it. The string is reversed out (white on black) when it is selected.

2. On the Model Maker viewer menu bar, select **Text | Size | 24**.

   The text string is redisplayed at the new point size. If the text overwrites any of the graphics in the model, you can simply click on it to select it and then drag it to a new location.

3. In the Model Maker viewer menu bar, select **Text | Style | Bold**.

The text string is redisplayed in bold type.

NOTE: If you want to edit a line of text, simply double-click it to bring up the Text String dialog again. Correct your entry or type a new one.

Add Text to a Function Graphic

1. In the Model Maker tool palette, select the Text tool and then the Lock tool to add text to the first Function graphic.
2. Click on the center of the **CONDITIONAL** Function graphic, toward the top of the graphic.

   The Text String dialog opens.
3. Type the following words in the Text String dialog:

   Create a sensitivity file by
4. Press Return on your keyboard, and then click **OK** in the Text String dialog.
5. Click under the first line of text to add another line.
6. In the Text String dialog, type:

   combining Slope, Floodplain, and Landcover
7. Press Return on your keyboard and then click **OK** in the Text String dialog.
8. Repeat step 5. to add a third line of text:

   using a conditional statement.
9. Click **OK**.

   All three text strings display over the Function graphic, but they are very large.

Format Text

1. In the Model Maker tool palette, click on the Lock icon to disable the lock tool and then click on the Select icon.
2. Click on the first line on the Function graphic to select it.
3. Shift-click on the second and third lines to add to the selection.
4. Using the same procedure you used to change the point size and style of the title, change these lines to **14** points, **Normal**.

   You may also want to adjust the positioning (simply drag on the text).

Add Text to Other Graphics

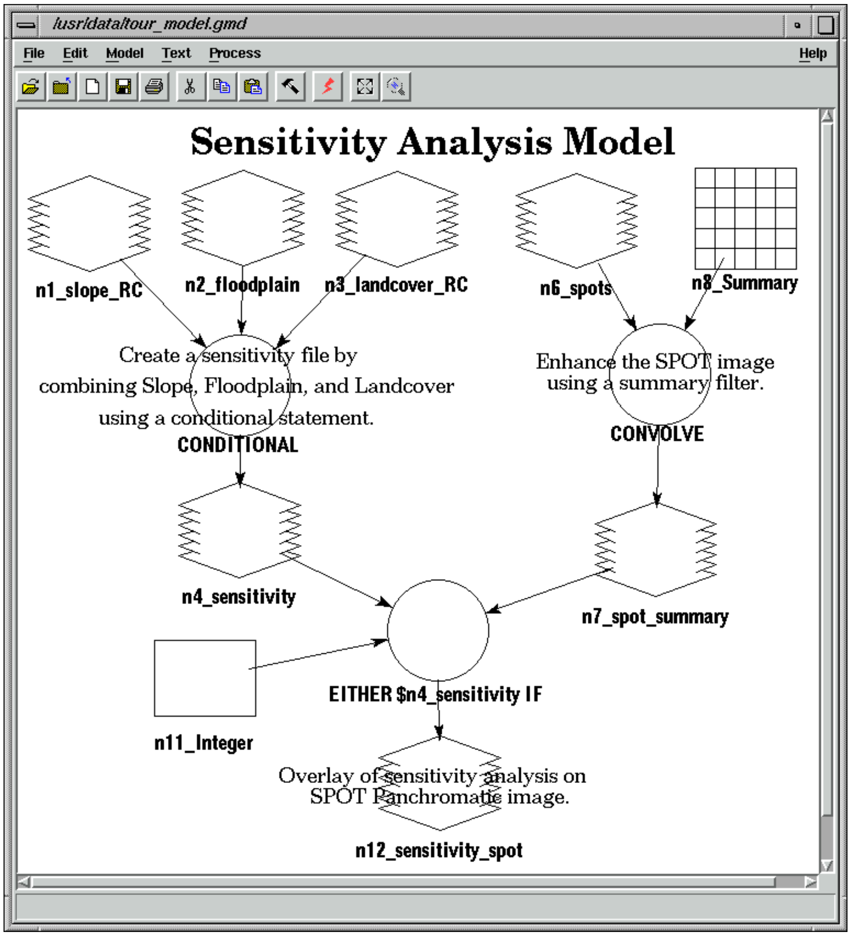
1. Add the following lines of text to the **CONVOLVE** function:

Enhance the SPOT image
using a summary filter.

2. Next, add these two lines to the final output raster (**n12_sensitivity_spot**):

Overlay of sensitivity analysis on
SPOT Panchromatic image.

Your annotated model should look like the following example.


/usr/data/tour_model.gmd
File
Edit
Model
Text
Process
Help
n1_slope_RC
n2_floodplain
n3_landcover_RC
n6_spots
n8_Summary
CONDITIONAL
CONVOLVE
n4_sensitivity
n7_spot_summary
EITHER $n4_sensitivity IF
n11_Integer
n12_sensitivity_spot

Sensitivity Analysis Model

Create a sensitivity file by
combining Slope, Floodplain, and Landcover
using a conditional statement.

Enhance the SPOT image
using a summary filter.

Overlay of sensitivity analysis on
SPOT Panchromatic image.

3. Save the model by selecting **File | Save** from the menu bar.

Generate a Text Script

The graphical models created in Model Maker can be output to a script file (text) in SML. Select **Tools | Edit Text Files** from the ERDAS IMAGINE menu bar, and then edit these scripts using the SML syntax. Re-run or save the edited scripts in the script library.

SML is designed for advanced modeling, and encompasses all of the functions available in Model Maker, as well as:

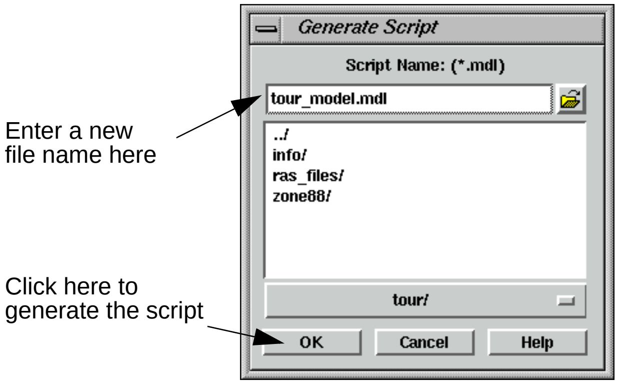
- conditional branching and looping
- complex data types
- flexibility in using raster objects

To generate a script from a graphical model, follow these steps:

The graphical model must be open.

1. In the Model Maker viewer menu bar, select **Process | Generate Script**.

   The Generate Script dialog opens.


Enter a new
file name here
Click here to
generate the script
Generate Script
Script Name: (*.mdl)
tour_model.mdl
../
info/
ras_files/
zone88/
tour/
OK
Cancel
Help

The **Script Name** defaults to the same root name as the graphical model. Scripts have the extension .mdl.

2. If you do not want to use the default, enter a new file name under **Script Name**.
3. Click **OK** to generate the script.

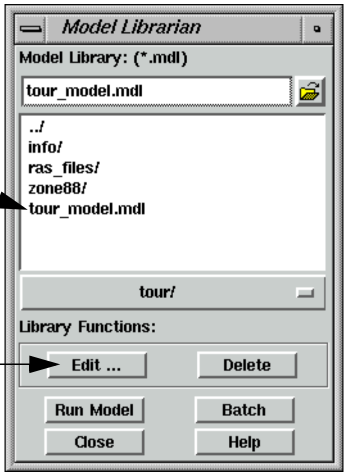
   The model is now accessible from the **Model Librarian** option of Spatial Modeler.

4. From the ERDAS IMAGINE icon panel, click the Modeler icon.

   The **Spatial Modeler** menu displays.

5. Select the **Model Librarian** option in the **Spatial Modeler** menu.

The Model Librarian dialog opens.


Model Librarian
Model Library: (*.mdl)
tour_model.mdl
../
info/
ras_files/
zone88/
tour_model.mdl
tour/
Library Functions:
Edit ...
Delete
Run Model
Batch
Close
Help

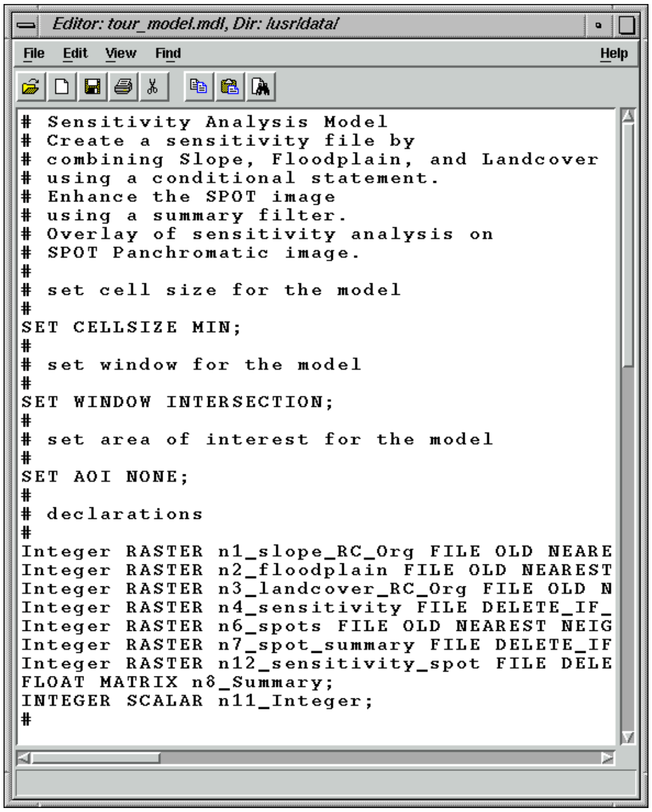
Click here to select the model

Click here to edit

From this dialog you can edit, delete, or run script models.

6. Under **Model Library**, select the name you used for your model in step 2.
7. Click **Edit** in the Model Librarian dialog.

The model is displayed in the Text Editor, as in the following example:



```
# Sensitivity Analysis Model
# Create a sensitivity file by
# combining Slope, Floodplain, and Landcover
# using a conditional statement.
# Enhance the SPOT image
# using a summary filter.
# Overlay of sensitivity analysis on
# SPOT Panchromatic image.
#
# set cell size for the model
#
SET CELLSIZE MIN;
#
# set window for the model
#
SET WINDOW INTERSECTION;
#
# set area of interest for the model
#
SET AOI NONE;
#
# declarations
#
Integer RASTER n1_slope_RC_Org FILE OLD NEARE
Integer RASTER n2_floodplain FILE OLD NEAREST
Integer RASTER n3_landcover_RC_Org FILE OLD N
Integer RASTER n4_sensitivity FILE DELETE_IF_
Integer RASTER n6_spots FILE OLD NEAREST NEIG
Integer RASTER n7_spot_summary FILE DELETE_IF
Integer RASTER n12_sensitivity_spot FILE DELE
FLOAT MATRIX n8_Summary;
INTEGER SCALAR n11_Integer;
#
```

Annotation in scripts is located at the top of the file, in the order in which it was entered. If you want the annotation to be in the order of processing, annotate your graphical model from top to bottom.

8. Select **File | Close** from the Text Editor menu bar.

9. Click **Close** in the Model Librarian dialog and the **Spatial Modeler** menu.

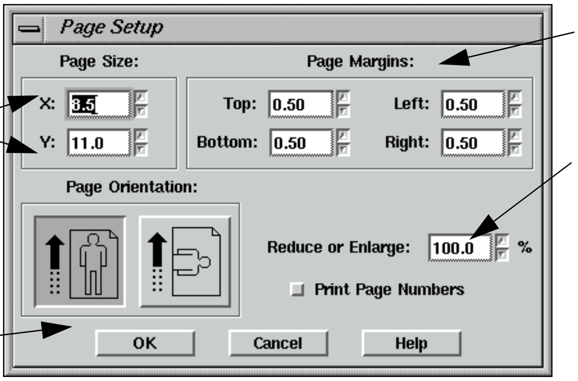
Print the Model

You can output graphical models as ERDAS IMAGINE annotation files (.ovr extension) and as encapsulated PostScript files (.eps extension). You can also output directly to a PostScript printer.

You must have a graphical model open.

1. In the Model Maker viewer menu bar, select **File | Page Setup**.

   The Page Setup dialog opens.



Page Size:
X: 8.5
Y: 11.0

Page Margins:
Top: 0.50
Left: 0.50
Bottom: 0.50
Right: 0.50

Reduce or Enlarge: 100.0 %

☐ Print Page Numbers

Indicate margins around the outside edges of each page here

Enter printer paper size here

Enter magnification or reduction to apply before printing here

Specify portrait or landscape printing here

The default setting specifies a 8.5” × 11” page size. This is acceptable for most PostScript printers.

2. In the Page Setup dialog, adjust the size of the **Page Margins** to suit your preferences.

3. Click **OK**.

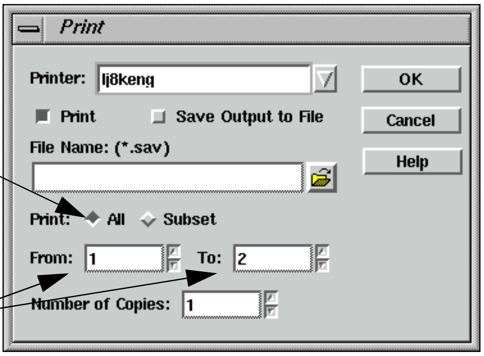
4. In the Model Maker viewer menu bar, select **File | Show Page Breaks**.

   Dotted lines indicate page breaks according to the page size specified in the Page Setup dialog. You may have to use the scroll bars on the bottom and side of the Model Maker viewer to see these page breaks.

5. If your model takes up more than one page, you may want to rearrange it so that it fits on a single sheet.

6. In the Model Maker viewer tool bar, click the Print icon (or select **File | Print** from the Model Maker viewer menu bar).

   The Print dialog opens.



Printer: lj8keng

■ Print ☐ Save Output to File

File Name: (*.sav)

Print: ◆ All ◇ Subset

From: 1 To: 2

Number of Copies: 1

Select to print the entire model

Select to print specific pages

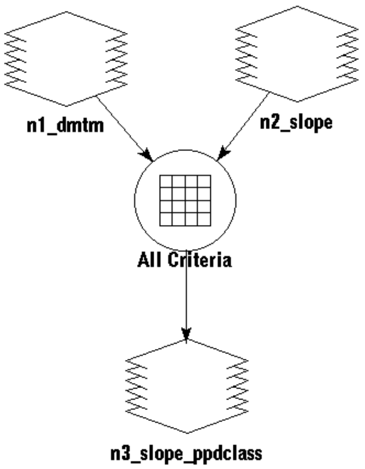
7. In the Print dialog, select the page(s) to print in the **From** and **To** boxes, or select **All** to print the entire model.
8. Click **OK** to print the model.

Apply the Criteria Function

The Criteria function in Model Maker simplifies the process of creating a conditional statement. In this example, you use data from a thematic raster layer and a continuous raster layer to create a new output layer. The input layers include a Landsat TM file and a slope file. This model performs similar to a parallelepiped classifier, but uses slope and image statistics in the decision process. The output file contains four classes: chaparral in gentle slopes, chaparral in steep slopes, riparian in gentle slopes, and riparian in steep slopes.

For information on the parallelepiped classifier, see "Chapter 17: Advanced Classification" *on page 455.*

The model you create looks like this:



Evaluate Training Samples

Before beginning, the ERDAS IMAGINE Classification tools were used to gather training samples of chaparral and riparian land cover. This was done to determine the minimum and maximum data file values of each class in three of the seven TM bands (4, 5, 3). These values are listed in the following table:

Table 16-3: Training Samples of Chaparral and Riparian Land Cover

Band	Chaparral		Riparian	
	Min	Max	Min	Max
4	31	67	55	92
5	30	61	57	87
3	23	37	27	40

Slopes below class value 3 are less than 8 percent, and therefore are characterized as gentle. Slopes in class value 3 or above are greater than 8 percent, and are characterized as steep. These values are used in the criteria function.

You must have Model Maker running, with a new Model Maker viewer displayed.

1. Click on the Raster icon in the Model Maker tool palette, then click the Lock icon.
2. Click three times in the Model Maker viewer to place the two input Raster graphics and the one output Raster graphic.
3. Click on the Criteria icon in the Model Maker tool palette.
4. Click in the Model Maker viewer to place the criteria graphic between the input and output Raster graphics.
5. Click on the Connect icon.
6. Connect the input Raster graphics to the criteria and the criteria to the output Raster graphic.
7. Click the Lock icon to disable the lock tool.
8. Click the Select icon.

Define Input Raster Layers

1. Double-click the first Raster graphic in the Model Maker viewer.

   The Raster dialog opens.

2. In the Raster dialog, click on the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, select the file **dmtm.img** and click **OK**.
4. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n1_dmtm** is written underneath the Raster graphic.

5. Double-click the second Raster graphic.

   The Raster dialog opens.

6. In the Raster dialog, click on the Open icon under **File Name**.

   The File Name dialog opens.

7. In the File Name dialog, select the file **slope.img** and click **OK**.
8. Click **OK** in the Raster dialog.

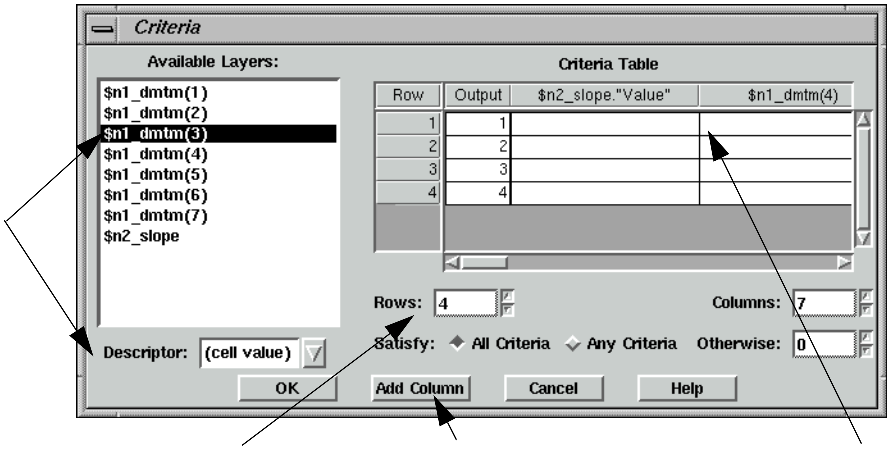
   The Raster dialog closes and **n2_slope** is written underneath the Raster graphic.

Define Criteria

1. Double-click the Criteria graphic in the Model Maker viewer.

   The Criteria dialog opens.

Click here to select a file, then click here to select a descriptor



$n1_dmtm(1)
$n1_dmtm(2)
$n1_dmtm(3)
$n1_dmtm(4)
$n1_dmtm(5)
$n1_dmtm(6)
$n1_dmtm(7)
$n2_slope

Descriptor: (cell value)

Row	Output	$n2_slope."Value"	$n1_dmtm(4)
1	1		
2	2		
3	3		
4	4		

Rows: 4

Columns: 7

Satisfy: ◆ All Criteria ◇ Any Criteria

Otherwise: 0

Change the number of rows here

Click here to add a column—columns display here

2. In the Criteria dialog, click on **$n2_slope** under **Available Layers**.

   The descriptor fields associated with that layer are now listed in the **Descriptor** popup list.

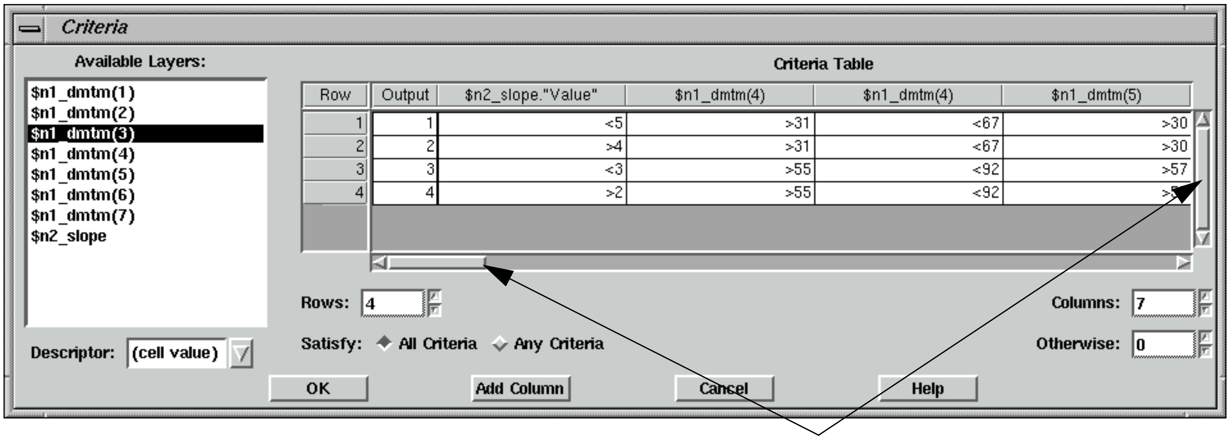
3. Click on the **Descriptor** popup list to select the **Value** descriptor.
4. Click **Add Column** to add that descriptor to the **Criteria Table**.
5. Under **Available Layers**, click on **$n1_dmtm(4)**, then click **Add Column** to add a column for the minimum data file values in band 4.
6. Click **Add Column** again to add a column for the maximum data file values in band 4.
7. Repeat this procedure for **$n1_dmtm(5)** and **$n1_dmtm(3)**.

   There are now eight columns in the **Criteria Table**.

8. Change the number of **Rows** to **4**, because the final output file has four classes.
9. Click in the first row of the **$n2_slope** column and type **<3**. Press Return on your keyboard.
10. Under **$n2_slope**, enter **>=3** in row **2**, **<3** in row **3**, and **>=3** in row **4**.
11. In the same manner, enter the minimum and maximum data file values for chaparral and riparian in the **Criteria Table**.

    Rows **1** and **2** correspond to chaparral, and rows **3** and **4** correspond to riparian (see the chart on page 376).

    The Criteria dialog should look like the one in the following diagram:



Criteria Table

Row	Output	$n2_slope."Value"	$n1_dmtm(4)	$n1_dmtm(4)	$n1_dmtm(5)
1	1	<5	>31	<67	>30
2	2	>4	>31	<67	>30
3	3	<3	>55	<92	>57
4	4	>2	>55	<92	>5

Descriptor: (cell value)

Rows: 4

Columns: 7

Satisfy: ◆ All Criteria ◇ Any Criteria

Otherwise: 0

Use the scroll bars to view all columns

The complete **Criteria Table** should look similar to the following table:

Table 16-4: Complete Criteria Table

Row	$n2_slope. "Value"	$n1_dmtm(4)	$n1_dmtm(4)	$n1_dmtm(5)	$n1_dmtm(5)	$n1_dmtm(3)	$n1_dmtm(3)
1	<3	>31	<67	>30	<61	>23	<37
2	>=3	>31	<67	>30	<61	>23	<37
3	<3	>55	<92	>57	<87	>27	<40
4	>=3	>55	<92	>57	<87	>27	<40

12. When all of the values are entered into the **Criteria Table**, click **OK**.

The Criteria dialog closes and **All Criteria** is written under the criteria graphic.

Define Output Raster Layer

1. Double-click the output Raster graphic in the Model Maker viewer.

The Raster dialog opens.

2. In the Raster dialog under **File Name**, enter the name **slope_ppdclass**, then Return on your keyboard.

Be sure that you specify a directory in which you have write permission.

3. Click on the **Delete if Exists** option.

4. Click on the **Data Type** popup list and select **Unsigned 4-bit**.

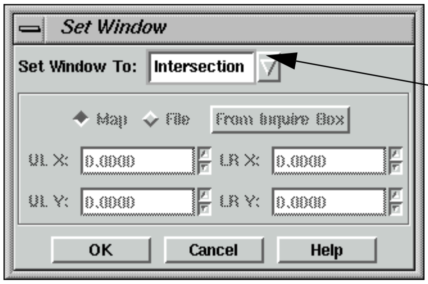
5. Confirm that **Thematic** is selected in the **File Type** popup list.

6. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n3_slope_ppdclass** is written under the output Raster graphic.

7. In the Model Maker viewer menu bar, select **Model | Set Window** to define the working window for the model.

   The Set Window dialog opens.



Set Window To: Intersection

Click on this popup
list to select Intersection

   You want the model to work on the intersection of the input files. The default setting is the union of these files.

8. In the Set Window dialog, click on the **Set Window To** popup list and select **Intersection**.

9. Click **OK** in the Set Window dialog.

Save the Model

1. Click the Save icon or select **File | Save As** from the Model Maker viewer tool bar to save your model.

   The Save Model dialog opens.

2. In the Save Model dialog, enter a name for your model.

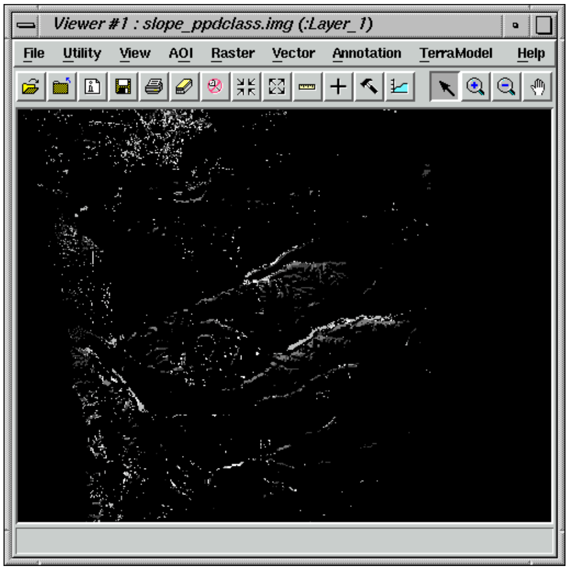
   Be sure you are saving the model in a directory in which you have write permission.

3. Click **OK** in the Save Model dialog.

4. In the Model Maker viewer tool bar, click the Run icon (or select **Process | Run** from the Model Maker viewer menu bar) to run the model.

   While the model runs, a Job Status dialog opens, reporting the status of the model.

5. When the model is finished, click **OK** in the Job Status dialog.
6. If you like, display **slope_ppdclass.img** in a Viewer to view the output image of your model.



The image displays in gray scale. The class values are defined in the criteria function where: 1—chaparral in gentle slopes, 2—chaparral in steep slopes, 3—riparian in gentle slopes, and 4—riparian in steep slopes.

Minimizing Temporary Disk Usage

The Spatial Modeler attempts to perform operations in memory where possible, but there are some common operations that produce temporary files. Any time a Global operation is performed on an intermediate result, a temporary file is produced. For example, if the Global Maximum pixel value is required for an image being calculated, nothing other than an estimate may be produced without actually generating the image.

If an intermediate image is going to be used in two or more additional functions in a model, a temporary file is created. Also if nonpoint functions like Spread and Clump are preformed on intermediate results, or if their results are used in further processes, temporary files are created.

There are two types of temporary files created by Spatial Modeler: temporary files, which are declared as such; and intermediate files, which get created due to the mix of operations. The amount of space required by temporary files can be controlled to some degree by user preferences. By default, Spatial Modeler is shipped to maintain the highest degree of precision at the expense of disk space. The default data type for both temporary and intermediate files is double precision floating point, which uses 8 bytes to store a pixel value. Depending on your needs, you can cut the size of your temporary files in half.

Set Preferences

1. Select **Session | Preferences** from the ERDAS IMAGINE menu bar.
2. In the Preference Editor, select the **Spatial Modeler** category.

3. Set **Float Temp Type** to **Single Precision Float**.
4. Set **Float Intermediate Type** to **Single Precision Float**.

Spatial Modeler, by default, also does not constrain the area your model processes, so temporary files extend to the union of all your input images. If, for example, you are doing an operation on two input images and your results are only valid in areas where both images exist, then setting the following preference may significantly reduce your temporary space requirements:

5. Set **Window Rule** to **Intersection**.

Also to ensure the temporary files get created on a disk drive where space is available, check the following preference:

6. In the Preference Editor, select the **User Interface & Session** category.
7. Set the **Temporary File Directory** to a local disk with sufficient free space.

In some cases, you may be able to adequately predict the output data range of a calculation. For example, if you calculate NDVI within your model, you know that at most, it can range from -0.5 to 0.5. In this case, you could

- store the result as floating point, taking at least 4 bytes per pixel, or
- scale the results to 0-255 in order to store the result as unsigned 8-bit data, taking just 1 byte per pixel. In this case since you know the range, you can re-scale the data by simply adding 0.5 then multiplying by 255, without the need for any temporary files.

For more extensive examples of how models may be written without the use of temporary disk space use Model Maker to open: 8bit_pca.gmd and 8bit_res_merge.gmd in the <IMAGINE_HOME>/etc/models directory, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

Making Your Models Usable by Others

Prompt User

When you specify specific input rasters or vectors in your model, their complete path is stored in the model. The same is true when you specify output files. So, to give someone else your models, they need to redefine all the inputs and outputs.

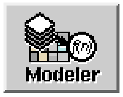
Starting with ERDAS IMAGINE 8.3, inputs and outputs can be set to **Prompt User** so that no absolute paths are contained in the model. The model, in turn, may easily be shared without the need to redefine any inputs or outputs.

Providing a User Interface to Your Model

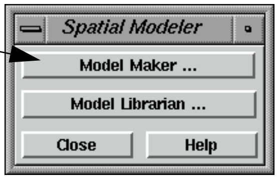
Another method of producing a model that can not only be easily shared with others, but is also very easy to run, is to write an EML front-end to your model.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon on the ERDAS IMAGINE icon panel.



The **Spatial Modeler** menu opens.


Spatial Modeler
Model Maker ...
Model Librarian ...
Close
Help

Select Model Maker

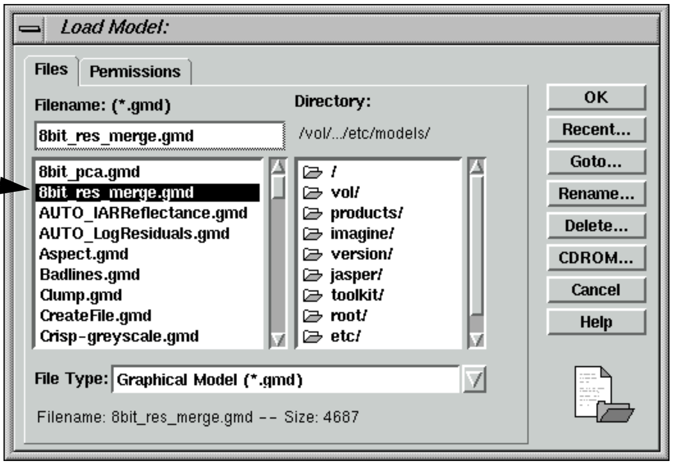
2. Click **Model Maker** on the **Spatial Modeler** menu.

   A blank Spatial Modeler viewer opens along with the Model Maker tool palette.

Open an Existing Model

1. Select **File | Open** or click on the Open Existing Model icon in the tool bar.

   The Load Model dialog opens.


Load Model:
Files
Permissions
8bit_pca.gmd
8bit_res_merge.gmd
AUTO_IARReflectance.gmd
AUTO_LogResiduals.gmd
Aspect.gmd
Badlines.gmd
Clump.gmd
CreateFile.gmd
Crisp-greyscale.gmd
/
vol/
products/
imagine/
version/
jasper/
toolkit/
root/
etc/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.gmd) 8bit_res_merge.gmd

Directory: /vol/.../etc/models/

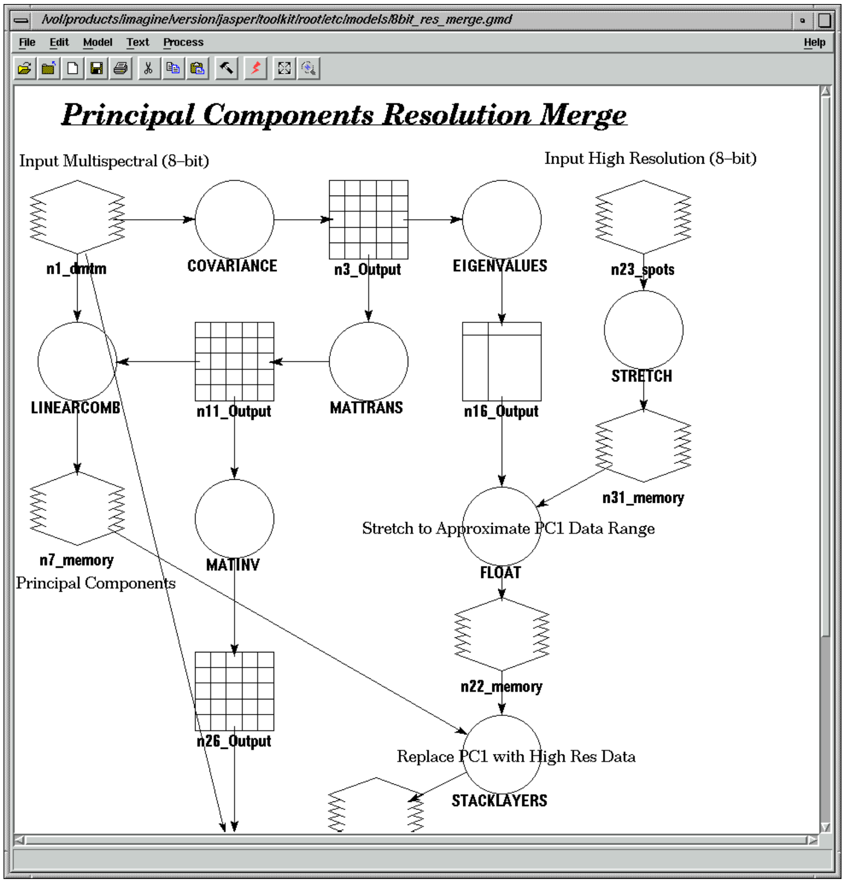
File Type: Graphical Model (*.gmd)

Filename: 8bit_res_merge.gmd -- Size: 4687

Select 8bit_res_merge.gmd from the list

2. Select **8bit_res_merge.gmd** in the Load Model dialog, and click **OK**.

   The model opens in the Spatial Modeler viewer.


/vol/products/imagine/version/jasper/toolkit/root/etc/models/8bit_res_merge.gmd
File
Edit
Model
Text
Process
Help
Principal Components Resolution Merge
Input Multispectral (8-bit)
Input High Resolution (8-bit)
n1_dmtm
COVARIANCE
n3_Output
EIGENVALUES
n23_spots
LINEARCOMB
n11_Output
MATTRANS
n16_Output
STRETCH
n31_memory
Stretch to Approximate PC1 Data Range
n7_memory
Principal Components
MATINV
FLOAT
n22_memory
n26_Output
Replace PC1 with High Res Data
STACKLAYERS

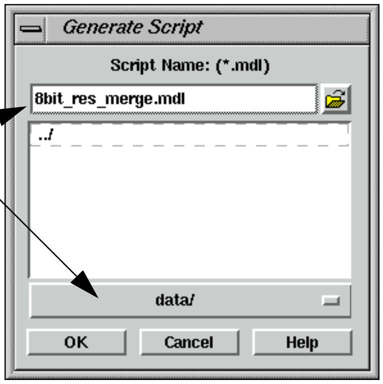
3. Select **Process | Generate Script**.

   The Generate Script dialog opens.

4. Click the Open icon in the Generate Script dialog, and navigate to a directory where you have write permission.

5. Name the file **8bit_res_merge.mdl**, then click **OK** in the dialog.

Save the .mdl file in a location where you have writer permission—you can then edit the file


Generate Script
Script Name: (*.mdl)
8bit_res_merge.mdl
../
data/
OK
Cancel
Help

Remember where you saved the file. You use it again on page 389.

6. Click **OK** in the Generate Script dialog.

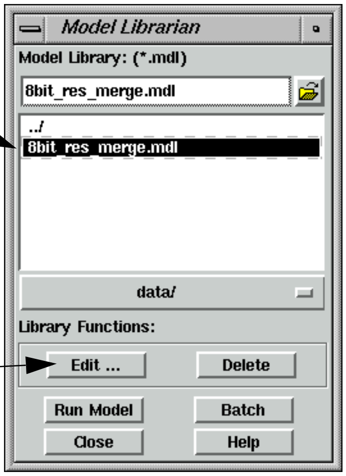
Edit the Model

1. In the **Spatial Modeler** menu, click on **Model Librarian**.

2. Navigate to the directly in which you saved **8bit_res_merge.mdl**, and select it.

Select the model from your writable directory

Click Edit to make changes to the model


Model Librarian
Model Library: (*.mdl)
8bit_res_merge.mdl
../
8bit_res_merge.mdl
data/
Library Functions:
Edit ...
Delete
Run Model
Batch
Close
Help

3. Click on the **Edit** button in the Model Librarian dialog.

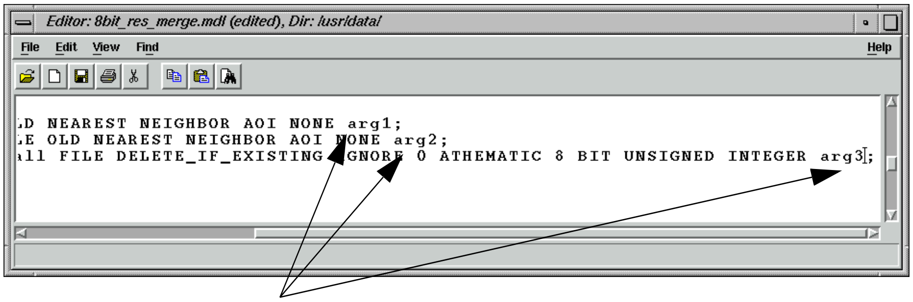
   The following SML script is displayed in the Editor:

```
# Principal Components Resolution Merge
# Input Multispectral (8-bit)
# Principal Components
# Replace PC1 with High Res Data
# Inverse PC
# Output Merged Image (8-bit)
# Input High Resolution (8-bit)
# Stretch to Approximate PC1 Data Range
#
# set cell size for the model
#
SET CELLSIZE MIN;
#
# set window for the model
#
SET WINDOW INTERSECTION;
#
# set area of interest for the model
#
SET AOI NONE;
#
# declarations
#
Float RASTER n1_dmtm FILE OLD NEAREST NEIGHBOR AOI NONE
            “$IMAGINE_HOME/examples/dmtm.img”;
Integer RASTER n23_spots FILE OLD NEAREST NEIGHBOR AOI NONE
            “$IMAGINE_HOME/examples/spots.img”;
Integer RASTER n29_merge_small FILE DELETE_IF_EXISTING IGNORE
            0 ATHEMATIC 8 BIT UNSIGNED INTEGER
            “c:/temp/merge_small.img”;
FLOAT MATRIX n3_Output;
FLOAT MATRIX n11_Output;
FLOAT MATRIX n26_Output;
FLOAT TABLE n16_Output;
{
#
# function definitions
#
#define n31_memory Float(STRETCH ($n23_spots(1) , 3 , 0 , 255 ))
n3_Output = COVARIANCE ( $n1_dmtm );
n11_Output = MATTRANS ( EIGENMATRIX ($n3_Output) );
n26_Output = MATINV ( $n11_Output ) ;
#define n7_memory Float(LINEARCOMB ( $n1_dmtm - GLOBAL MEAN (
            $n1_dmtm ) , $n11_Output ))
n16_Output = EIGENVALUES ( $n3_Output ) ;
#define n22_memory Float(FLOAT((($n31_memory - 127.5) * 3 *
            (SQRT($n16_Output[0]))) / 127.5))
#define n38_memory Float(STACKLAYERS($n22_memory ,
            $n7_memory(2: NUMLAYERS ($n7_memory))))
n29_merge_small = LINEARCOMB ( $n38_memory , $n26_Output ) +
            GLOBAL MEAN ( $n1_dmtm );
}
QUIT;
```

4. Locate **“$IMAGINE_HOME/examples/dmtm.img”** (in bold above) on line 24 and change it to **arg1**.

5. Locate **“$IMAGINE_HOME/examples/spots.img”** (in bold above) on line 25 and change it to **arg2**.

6. Locate **“c:/temp/merge_small.img”** (in bold above) on line 26 and change it to **arg3**.



```
.D NEAREST NEIGHBOR AOI NONE arg1;
.E OLD NEAREST NEIGHBOR AOI NONE arg2;
all FILE DELETE_IF_EXISTING IGNORE 0 ATHEMATIC 8 BIT UNSIGNED INTEGER arg3;
```

The code now has arg1, arg2, and arg3

7. Select **File | Save**, or click on the Save Current Document icon in the Editor.

Edit the EML

1. Select **File | New**, or click on the New icon in the Editor.

2. Select **File | Open**, or click on the Open icon.

3. In the Load File dialog, type ***.eml** for the **File Name** and press Return on your keyboard.

   This searches for EML scripts in the directory.

4. Select **8bit_res_merge.eml**, and click **OK** in the Load File directory.

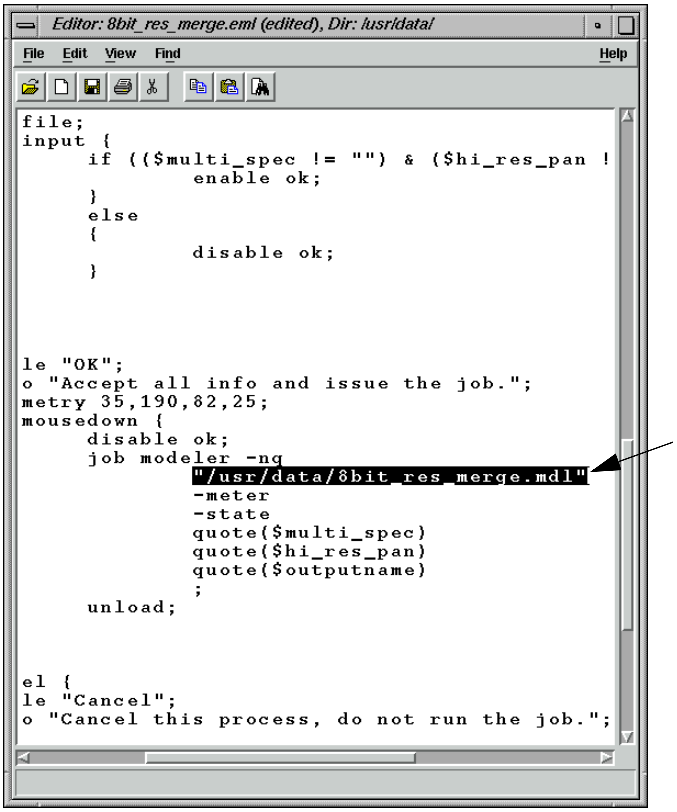
   The .eml file is located in <IMAGINE_HOME>/scripts, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

   The following EML script displays in the Editor:

```
component res_merge {

frame res_merge {
              title “Resolution Merge”;
              geometry 140,120,250,230;
              statusbar;
              filename hi_res_pan;
              filename outputname;
              button ok;
              filename multi_spec {
                  title above center “Multispectral File:”;
                  info “Select the multispectral input file.”;
                  shortform;
                  geometry 0,10,245,49;
                  select getpref (“eml” “default_data_path”)+”/*.img”;
                  filetypedef “raster”;
                  on input {
                      if (($multi_spec != ““) & ($hi_res_pan != ““) &
                        ($outputname != ““)) { enable ok;
                      }
                      else
                      {
                        disable ok;
                      }
                  }

              }

                  filename hi_res_pan {
                      title above center “High Resolution Pan File:”;
                      info “Select the high resolution pan input file.”;
                      shortform;
                      geometry 0,70,245,49;
                      select getpref (“eml” “default_data_path”)+”/*.img”;
                      filetypedef “raster”;
                      on input {
                        if (($multi_spec != ““) & ($hi_res_pan != ““) &
                          ($outputname != ““)) { enable ok;
                        }
                        else
                        {
                          disable ok;
                        }
                      }
                  }

              filename outputname {
                  title above center “Output File:”;info “Select output file.”;
                  shortform;
                  geometry 0,130,245,49;
                  select getpref (“eml” “default_output_path”)+”/*.img”;
                  filetypedef “raster” pseudotypes off creatable on;
                  newfile;
                  on input {
                      if (($multi_spec != ““) & ($hi_res_pan != ““) &
                        ($outputname != ““)) { enable ok;
                      }
```

```
                              else
                              {
                                 disable ok;
                              }
                        }
                  }

                        button ok {
                              title “OK”;
                              info “Accept all info and issue the job.”;
                              geometry 35,190,82,25;
                              on mousedown {
                                 disable ok;
                                 job modeler -nq
                                    “d:/erdas/models/8bit_res_merge.mdl”
                                    -meter
                                    -state
                                    quote($multi_spec)
                                    quote($hi_res_pan)
                                    quote($outputname)
                                    ;
                                 unload;

                              }

                        }

                        button cancel {
                              title “Cancel”;
                              info “Cancel this process, do not run the job.”;
                              geometry 140,190,82,25;
                              on mousedown {
                                 unload ;
                              }
                        }

                        on framedisplay {
                              disable ok;
                        }
}

on startup {
                        display res_merge;
}
}
```

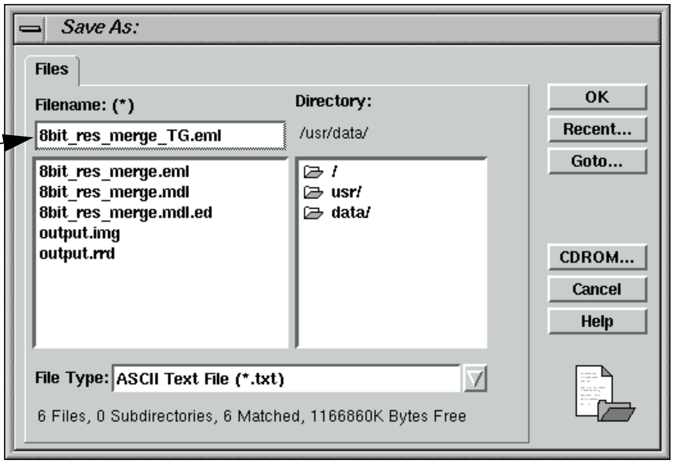
5. Locate **“d:/erdas/models/8bit_res_merge.mdl”** (in bold above) on line 74, and change it to the location and name of the script you generated.


Editor: 8bit_res_merge.eml (edited), Dir: /usr/data/
File
Edit
View
Find
Help

```
file;
input {
      if (($multi_spec != "") & ($hi_res_pan !
              enable ok;
      }
      else
      {
              disable ok;
      }



le "OK";
o "Accept all info and issue the job.";
metry 35,190,82,25;
mousedown {
      disable ok;
      job modeler -nq
              "/usr/data/8bit_res_merge.mdl"
              -meter
              -state
              quote($multi_spec)
              quote($hi_res_pan)
              quote($outputname)
              ;
      unload;



el {
le "Cancel";
o "Cancel this process, do not run the job.";
```

Change this section of the code to reflect the location of the .mdl file you edited with arg1, arg2, and arg3

6. Select **File | Save As**.


Save As:
Files
OK
Recent...
Goto...
CDROM...
Cancel
Help

Filename: (*) 8bit_res_merge_TG.eml

Directory: /usr/data/

8bit_res_merge.eml
8bit_res_merge.mdl
8bit_res_merge.mdl.ed
output.img
output.rrd

/
usr/
data/

File Type: ASCII Text File (*.txt)

6 Files, 0 Subdirectories, 6 Matched, 1166860K Bytes Free

Save the .eml file with a new name in a directory where you have write permission

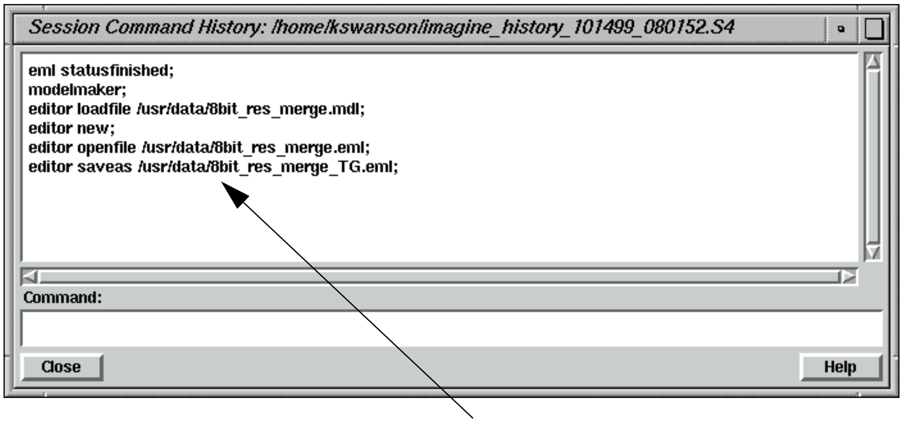
7. In the Save As dialog, navigate to a directory where you have write permission.

8. Save the .eml file as **8bit_res_merge_TG.eml**, then click **OK** in the Save As dialog.

Set Session Commands

1. On the ERDAS IMAGINE menu bar select **Session | Commands**.

   The Session Command dialog opens.

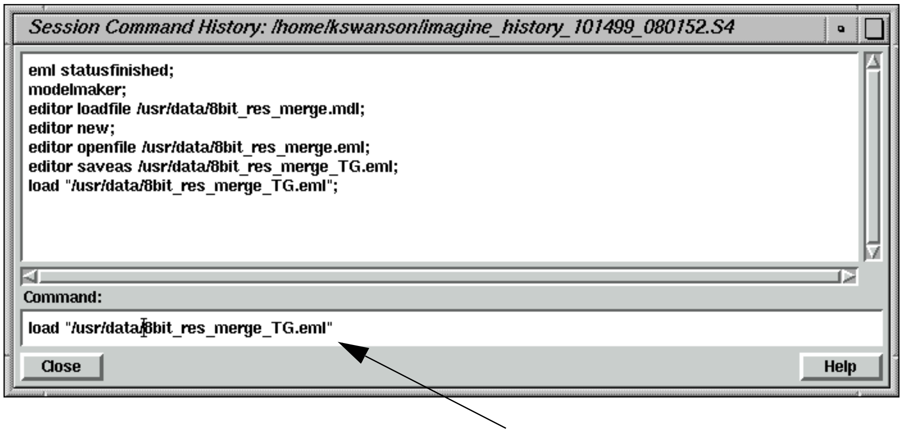

Session Command History: /home/kswanson/imagine_history_101499_080152.S4
Command:
Close
Help

```
eml statusfinished;
modelmaker;
editor loadfile /usr/data/8bit_res_merge.mdl;
editor new;
editor openfile /usr/data/8bit_res_merge.eml;
editor saveas /usr/data/8bit_res_merge_TG.eml;
```

Your most recent commands are listed here

2. In the **Command** field enter the following command (replacing the directory with the one you chose).

```
load “c:/temp/8bit_res_merge.eml”
```


Session Command History: /home/kswanson/imagine_history_101499_080152.S4
Command:
Close
Help

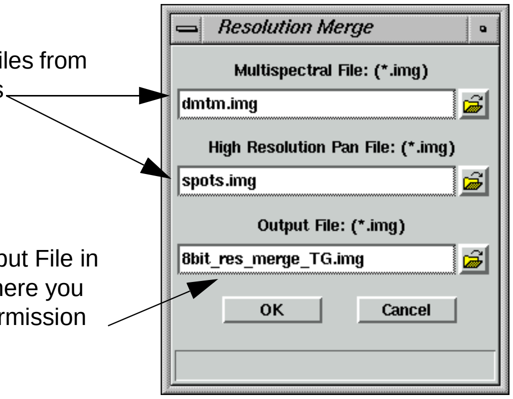
```
eml statusfinished;
modelmaker;
editor loadfile /usr/data/8bit_res_merge.mdl;
editor new;
editor openfile /usr/data/8bit_res_merge.eml;
editor saveas /usr/data/8bit_res_merge_TG.eml;
load "/usr/data/8bit_res_merge_TG.eml";
```

Command: `load "/usr/data/8bit_res_merge_TG.eml"`

Type the load command, plus the location of the .eml file here to execute the model

3. Press Return on your keyboard.

   The following dialog displays:



Select these files from the /examples directory

Save the Output File in a directory where you have write permission

Resolution Merge

Multispectral File: (*.img)
dmtm.img

High Resolution Pan File: (*.img)
spots.img

Output File: (*.img)
8bit_res_merge_TG.img

4. For the **Multispectral File**, select **dmtm.img** from the /examples directory.

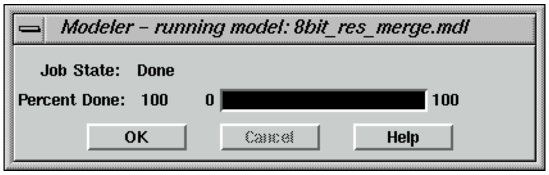
   The file **dmtm.img** *is located in the <IMAGINE_HOME>/examples directory, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.*

5. For the **High Resolution Pan File**, select **spots.img** from the examples directory.

6. For the **Output File**, select a directory in which you have write permission, and enter the name **8bit_res_merge_TG.img**, then press Return on your keyboard.

7. Click **OK**.

   A Job Status dialog opens, tracking the progress.



Modeler – running model: 8bit_res_merge.mdl

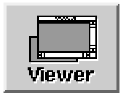
Job State: Done
Percent Done: 100

8. When the job is 100% complete, click **OK** in the dialog.

   You can set the **Keep Job Status Box** *in the* **User Interface & Session** *category of the Preference Editor so that the Job Status box is dismissed automatically after an operation is performed.*

Check the Results

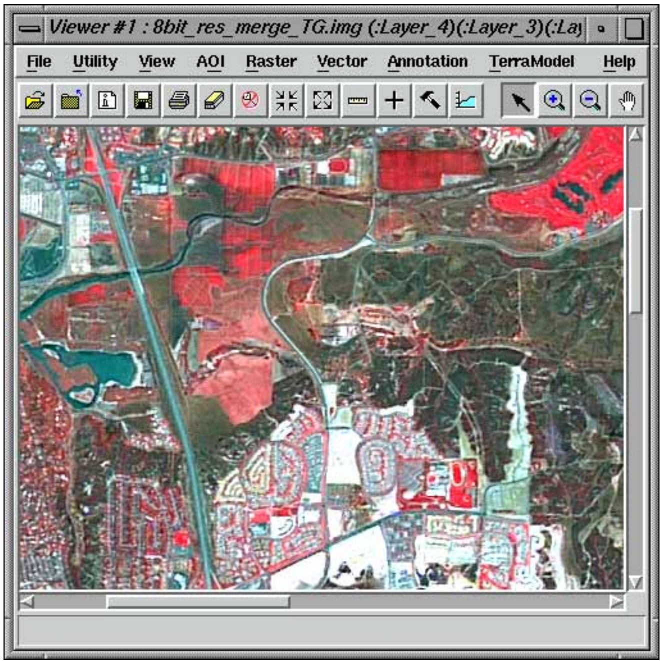
1. In the ERDAS IMAGINE icon panel, click the Viewer icon.



2. Click the Open icon, and navigate to the directory in which you saved the **Output File** you just created, **8bit_res_merge_TG.img**

3. Click **OK** in the Select Layer To Add dialog to add the file.

   The image displays in the Viewer.


Viewer #1 : 8bit_res_merge_TG.img (:Layer_4)(:Layer_3)(:La
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

4. Click the Open icon, and navigate to the /examples directory.

5. Select the file **dmtm.img** from the list, then click the **Raster Options** tab.

6. Deselect the **Clear Display** option.

7. Click **OK** in the Select Layer To Add dialog.

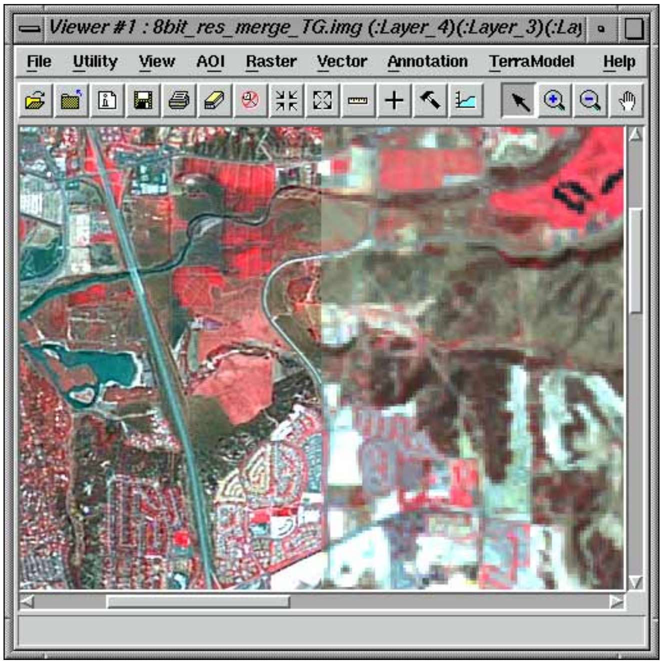
   The multispectral file, **dmtm.img**, lends the color to the resulting file: **8bit_res_merge_TG.img**.

Use the Swipe Utility

1. From the Viewer menu bar, select **Utility | Swipe**.

2. Move the slider bar back and forth to see how the two images compare.

3. When you are finished, close the Swipe utility.

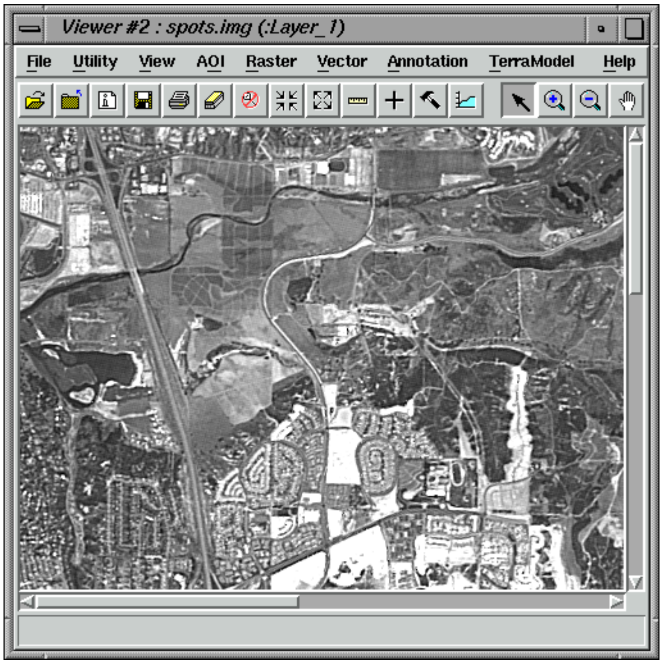

Viewer #1 : 8bit_res_merge_TG.img (:Layer_4)(:Layer_3)(:La
File Utility View AOI Raster Vector Annotation TerraModel Help

Check the spots.img image

The panchromatic image, **spot.img**, is the image that lends the detail to the image you created: **8bit_res_merge_TG.img**.

1. Click the Viewer icon on the ERDAS IMAGINE icon panel to open a new Viewer.
2. Click the Open icon, and navigate to the /examples directory in the Select Layer To Add dialog.
3. Select the file **spots.img**, then click **OK** in the Select Layer To Add dialog.

The file **spots.img** displays in the Viewer. Note the detail in the image.



4. When you are finished evaluating the images, select **Session | Close All Viewers** from the ERDAS IMAGINE menu bar.

5. Close the editors.

6. Save changes to your .eml file, **8bit_res_merge_TG.eml**.

7. Close the .gmd file, do not save changes.

Using Vector Layers in Your Model

Vector layers may be used in several different ways within models. All processing is done in raster format. However, converting the vector layers to raster is done on the fly at either a default resolution or one specified to meet the level of detail required by the application.

Vector Layers as a Mask

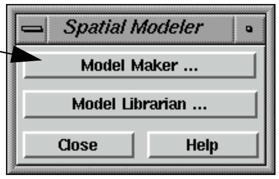
One simple application of vector layers is to use polygonal boundaries to cookie-cut your imagery. Whether the polygons represent political boundaries, ownership boundaries, zoning, or study area boundaries, they may be used to limit your analysis to just the portions of the imagery of interest.

In the following example you use a vector coverage to not just cookie-cut an image but to generate an output image for visual presentation that highlights the study area. Inside the study area, you enhance the image, while outside the study area you blur the image to further distinguish the study area.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon on the ERDAS IMAGINE icon panel.

The **Spatial Modeler** menu opens.



Select Model Maker

2. Click on the **Model Maker** button in the **Spatial Modeler** menu.

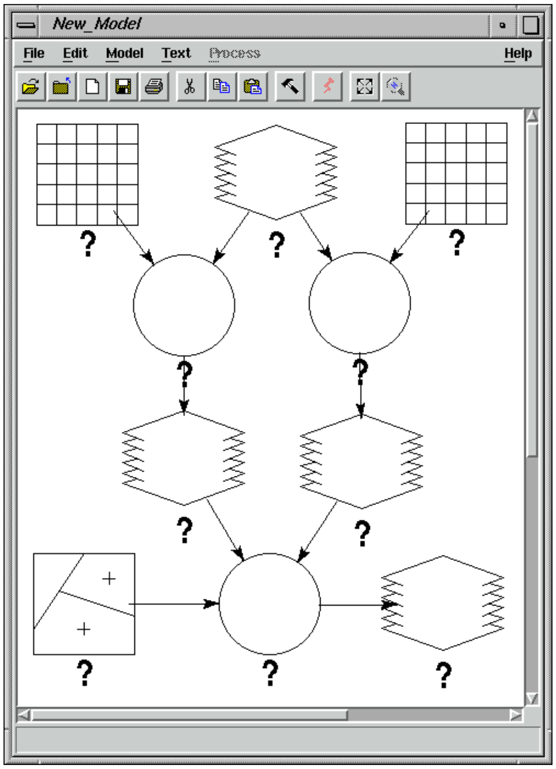
   A blank Model Maker viewer opens along with tools.

Set up the Model

1. Click on the Raster icon in the Model Maker tool palette.
2. Click near the top center of the Model Maker viewer.
3. Click on the Matrix icon in the Model Maker tool palette.
4. Click to the left of the Raster object in the Model Maker viewer.
5. Click on the Matrix icon again in the Model Maker tool palette.
6. Click to the right of the Raster object in the Model Maker viewer.

7. Click on the Function icon in the Model Maker tool palette.
8. Click below and to the left of the Raster object in the Model Maker viewer.
9. Click on the Function icon again in the Model Maker tool palette.
10. Click below and to the right of the Raster object in the Model Maker viewer.
11. Click on the Raster icon in the Model Maker tool palette.
12. Click below the first Function object in the Model Maker viewer.
13. Click on the Raster icon again in the Model Maker tool palette.
14. Click below the second Function object in the Model Maker viewer.
15. Click on the Function icon in the Model Maker tool palette.
16. Click below and between the Raster objects just placed in the Model Maker viewer.
17. Click on the Vector icon in the Model Maker tool palette.
18. Click to the left of the Function object just placed in the Model Maker viewer.
19. Click on the Raster icon in the Model Maker tool palette.
20. Click to the right of the Function object just placed in the Model Maker viewer.
21. Using the Connection tool, and optionally the Lock tool, connect the objects in the model as depicted in the following picture:

    When you are finished, the model looks like the following:

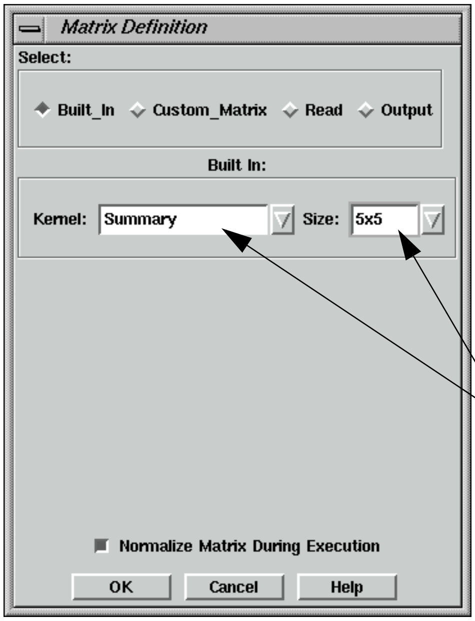

New_Model
File
Edit
Model
Text
Process
Help
?

Add Matrix Properties

1. Make sure the Selector tool is active.

2. Double-click the top left Matrix object in the Model Maker viewer.

   The Matrix Definition and Matrix dialogs open.



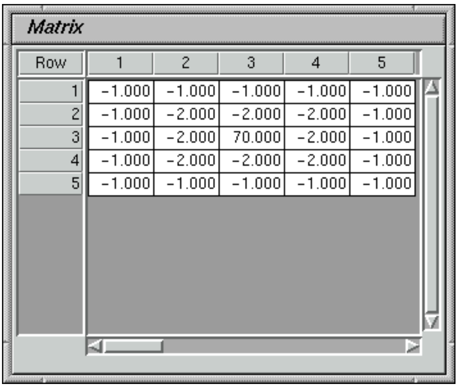
Select:

◆ Built_In ◇ Custom_Matrix ◇ Read ◇ Output

Built In:

Kernel: Summary Size: 5x5

■ Normalize Matrix During Execution



Row	1	2	3	4	5
1	-1.000	-1.000	-1.000	-1.000	-1.000
2	-1.000	-2.000	-2.000	-2.000	-1.000
3	-1.000	-2.000	70.000	-2.000	-1.000
4	-1.000	-2.000	-2.000	-2.000	-1.000
5	-1.000	-1.000	-1.000	-1.000	-1.000

Select the 5 × 5 Summary Kernel

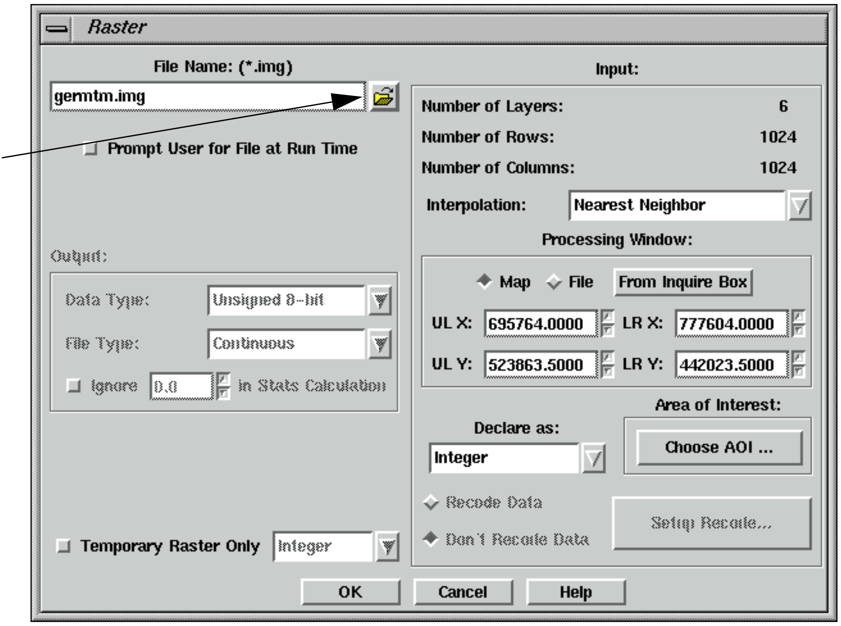
3. Using the **Kernel** popup list select **Summary**.

4. Using the **Size** popup list select **5x5**.

5. Click the **OK** button in the Matrix Definition dialog.

Add Raster Properties

1. Double-click the top Raster object in the Model Maker viewer.



Select the file here

Raster

File Name: (*.img) germtm.img

☐ Prompt User for File at Run Time

Output:

Data Type: Unsigned 8-bit

File Type: Continuous

☐ Ignore 0.0 in Stats Calculation

☐ Temporary Raster Only Integer

Input:

Number of Layers:	6
Number of Rows:	1024
Number of Columns:	1024

Interpolation: Nearest Neighbor

Processing Window:

● Map ○ File From Inquire Box

UL X: 695764.0000 LR X: 777604.0000

UL Y: 523863.5000 LR Y: 442023.5000

Declare as: Integer

Area of Interest: Choose AOI ...

○ Recode Data

● Don't Recode Data

Setup Recode...

OK Cancel Help

2. Click the Open icon to open the File Name dialog.

3. Select **germtm.img** from the examples directory, and click **OK** in the File Name dialog.

4. Click **OK** in the Raster dialog to accept the file **germtm.img**.

Add Matrix Properties

1. Double-click the top right Matrix object in the Model Maker viewer.

2. Verify that **Low Pass** is selected in the **Kernel** popup list.

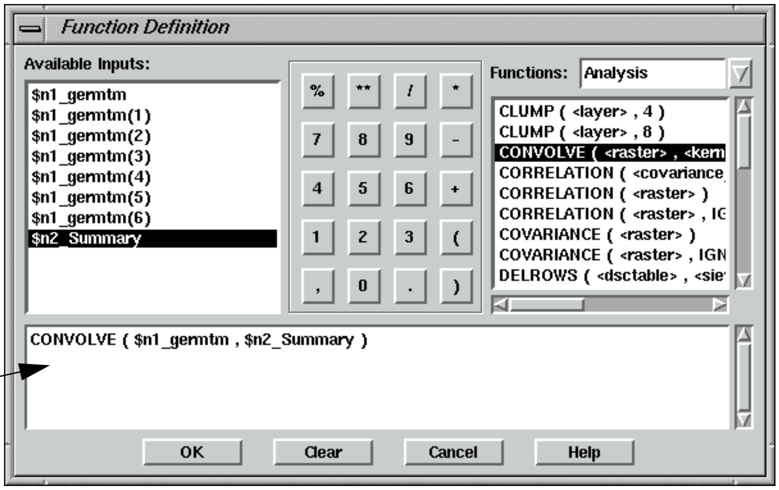
3. Using the **Size** popup list select **7x7**.

4. Click the **OK** button in the Matrix Definition dialog.

Add Function Properties

1. Double-click the left Function object.

The Function Definition dialog opens.



Available Inputs:

$n1_germtm
$n1_germtm(1)
$n1_germtm(2)
$n1_germtm(3)
$n1_germtm(4)
$n1_germtm(5)
$n1_germtm(6)
$n2_Summary

Functions: Analysis

CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie

CONVOLVE ($n1_germtm , $n2_Summary)

Define the Analysis Function using Inputs in this window

2. From the Analysis Functions select **CONVOLVE (<raster> , <kernel>)**, this should be the third item on the list.

3. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

4. Under Available Inputs, click on **$n1_germtm**.

5. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.

6. Under Available Inputs, click on **$n2_Summary**.

7. Click **OK** in the Function Definition dialog.

Add Function Properties

1. Double-click the right Function object.

2. From the Analysis Functions select **CONVOLVE (<raster> , <kernel>)**, this should be the third item on the list.

3. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

4. Under Available Inputs, click on **$n1_germtm**.

5. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.

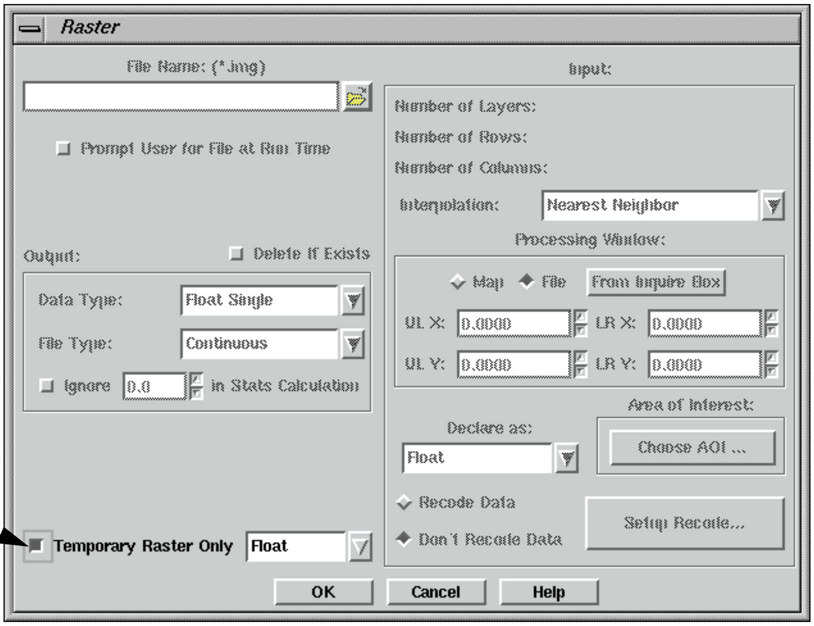
6. Under Available Inputs, click on **$n3_Low_Pass**.

   Your function string should look like the following:
   CONVOLVE ($n1_germtm , $n3_Low_Pass)

7. Click **OK** in the Function Definition dialog.

Add Raster Properties

1. Double-click the Raster object that is output from the Function on the left.



Raster
File Name: (*.img)
Prompt User for File at Run Time
Output:
Delete If Exists
Input:
Number of Layers:
Number of Rows:
Number of Columns:
Processing Window:
Map
File
From Inquire Box
Area of Interest:
Choose AOI ...
Recode Data
Don't Recode Data
Setup Recode...
Temporary Raster Only
OK
Cancel
Help

Data Type: Float Single

File Type: Continuous

☐ Ignore 0.0 in Stats Calculation

Interpolation: Nearest Neighbor

UL X: 0.0000 LR X: 0.0000

UL Y: 0.0000 LR Y: 0.0000

Declare as: Float

Temporary Raster Only: Float

Click this checkbox

2. In the lower left corner of the dialog click the checkbox **Temporary Raster Only**.
3. Click **OK** in the Raster dialog.

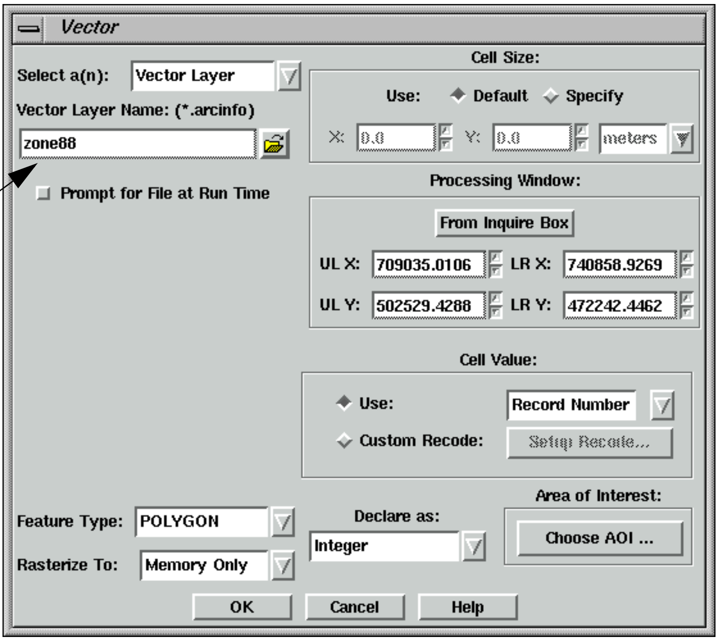
Add Raster Properties

1. Double-click the Raster object that is output from the Function on the right.
2. In the lower left corner of the dialog click the checkbox **Temporary Raster Only**.
3. Click **OK** in the Raster dialog.

Add Vector Properties

1. Double-click the Vector object in the lower left corner of the model.

   The Vector dialog opens.



Select a(n): Vector Layer
Vector Layer Name: (*.arcinfo)
zone88
☐ Prompt for File at Run Time
Cell Size:
Use: ● Default ○ Specify
X: 0.0 Y: 0.0 meters
Processing Window:
UL X: 709035.0106 LR X: 740858.9269
UL Y: 502529.4288 LR Y: 472242.4462
Cell Value:
● Use: Record Number
○ Custom Recode:
Feature Type: POLYGON
Rasterize To: Memory Only
Declare as: Integer
Area of Interest:

Select zone88 in the /examples directory

2. Click the Open icon under **Vector Layer Name**.

3. In the Vector Layer Name dialog, navigate to the /examples directory, and select **zone88**.

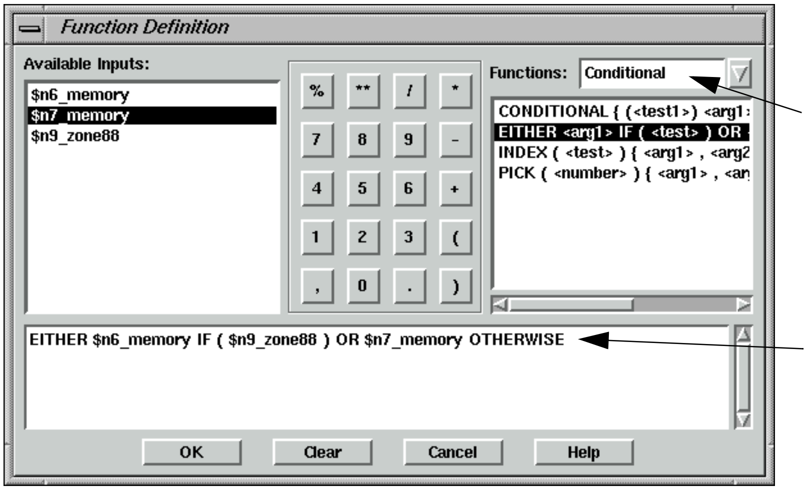
4. Click **OK** in the Vector Layer Name dialog.

5. Click **OK** in the Vector dialog to accept the **Vector Layer Name**.

Add Function Properties

1. Double-click the final Function object.

   The Function Definition dialog opens.



Available Inputs:
$n6_memory
$n7_memory
$n9_zone88

Functions: Conditional

CONDITIONAL { (<test1>) <arg1>:
EITHER <arg1> IF (<test>) OR
INDEX (<test>) { <arg1> , <arg2
PICK (<number>) { <arg1> , <ar

EITHER $n6_memory IF ($n9_zone88) OR $n7_memory OTHERWISE

Select the Conditional Function

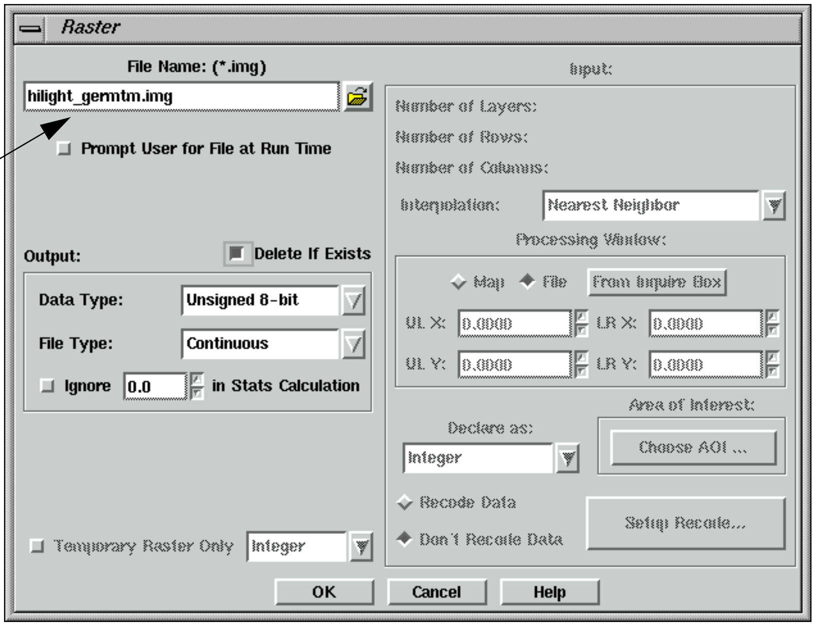
Edit the Function here

2. In the **Functions** popup list, select **Conditional**.
3. In the list of **Functions**, select **EITHER <arg1> IF (<test>) OR <arg2> OTHERWISE**, this should be the second item on the list.
4. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.
5. Under **Available Inputs**, click on **$n6_memory**.
6. In the lower portion of the Function Definition dialog, click in the middle of **<test>**.
7. Under **Available Inputs**, click on **$n9_zone88**.
8. In the lower portion of the Function Definition dialog, click in the middle of **<arg2>**.
9. Under **Available Inputs**, click on **$n7_ memory**.
10. Click **OK** in the Function Definition dialog.

Add Raster Properties

1. Double-click the final output Raster object.

   The Raster dialog opens.



Raster
Input:
Number of Layers:
Number of Rows:
Number of Columns:
Processing Window:
Area of Interest:
From Inquire Box
Choose AOI ...
Setup Recode...
OK
Cancel
Help

The output file name is listed here

File Name: (*.img) hilight_germtm.img

☐ Prompt User for File at Run Time

Output: ☑ Delete If Exists

Data Type: Unsigned 8-bit

File Type: Continuous

☐ Ignore 0.0 in Stats Calculation

☐ Temporary Raster Only Integer

Interpolation: Nearest Neighbor

○ Map ● File

UL X: 0.0000 LR X: 0.0000

UL Y: 0.0000 LR Y: 0.0000

Declare as: Integer

○ Recode Data

● Don't Recode Data

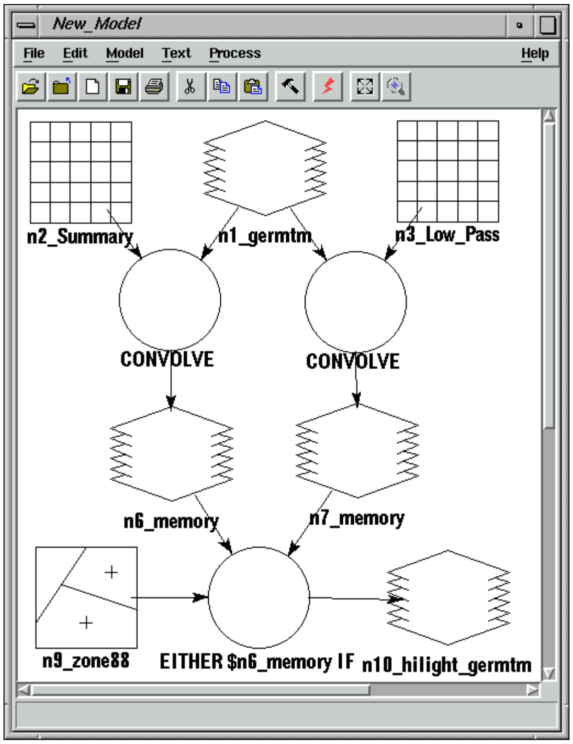
2. Click the Open icon and navigate to a directory where you have write permission.

3. In the **Filename** section of the File Name dialog, type **hilight_germtm.img** for the output image name, then click **OK** in the File Name dialog.

4. Click on **Delete If Exists** checkbox.

5. Click on the **OK** button in the Raster dialog.

   Your completed model should like the following:


New_Model
File
Edit
Model
Text
Process
Help
n2_Summary
n1_germtm
n3_Low_Pass
CONVOLVE
CONVOLVE
n6_memory
n7_memory
n9_zone88
EITHER $n6_memory IF
n10_hilight_germtm

Execute the Model and Check Results

1. Select **Process | Run**, or click on the Execute the Model icon in the tool bar.

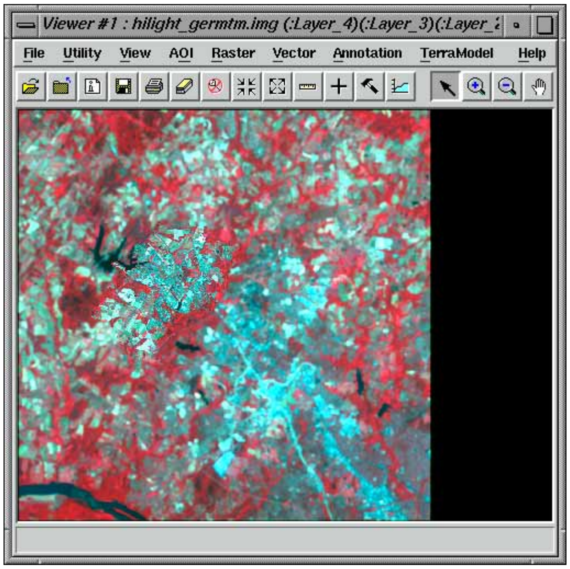
   A Job Status dialog opens tracking the progress of the function.

2. Click **OK** in the Job Status dialog when it reaches 100% complete.

   Next, use the Viewer to examine your output image and locate the highlighted area.

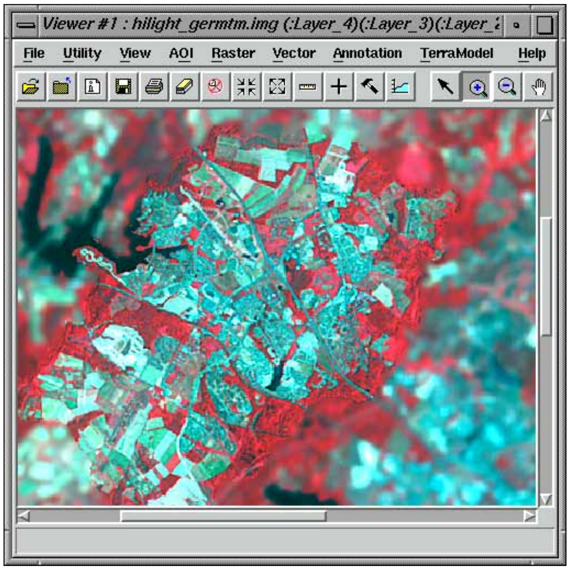
3. Click the Viewer icon in the ERDAS IMAGINE icon bar.

4. Click the Open icon and navigate to the directory in which you saved the output file, **hilight_germtm.img**, then click **OK** in the dialog to display the file.


Viewer #1 : hilight_germtm.img (:Layer_4)(:Layer_3)(:Layer_
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

5. Use the Zoom In tool to view the highlighted area.

Notice that the area you emphasized with the model is sharp, while the area surrounding it is fuzzy.


Viewer #1 : hilight_germtm.img (:Layer_4)(:Layer_3)(:Layer_
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

6. When you are finished viewing the image select **File | Close** from the Viewer menu bar.

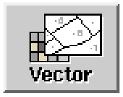
Add Attributes to Vector Layers

Another application of using vector layers in models is to calculate summary information about your imagery for each polygon in a vector layer. This summary information can then be stored as an additional attribute of the vector layer.

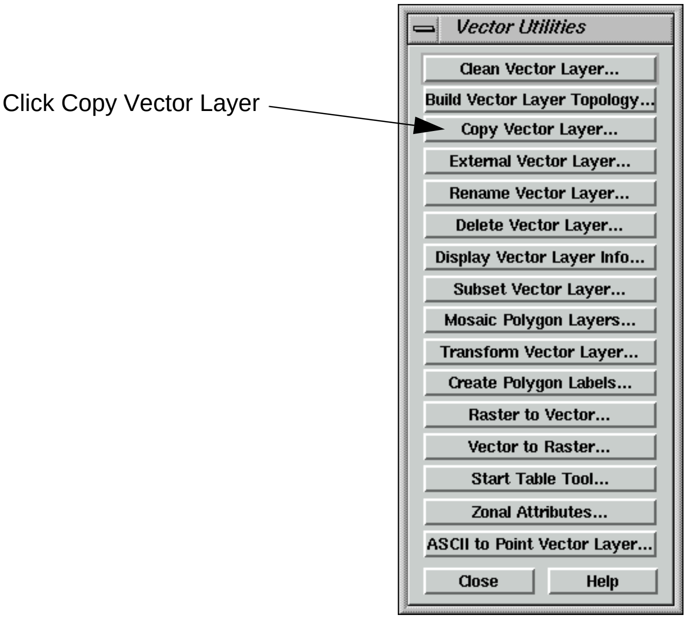
Copy Vector Layers

You must have ERDAS IMAGINE running.

1. Click the Vector icon on the ERDAS IMAGINE icon bar.

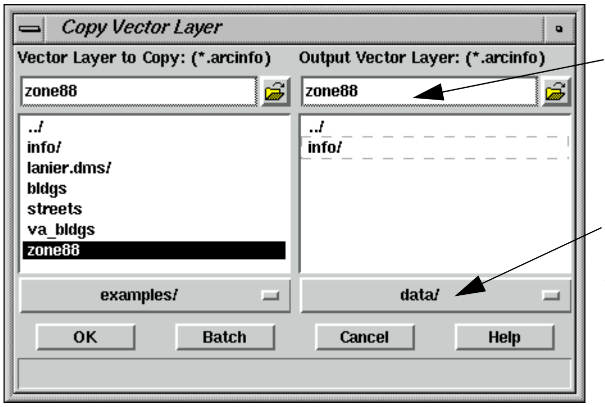


The **Vector Utilities** menu opens.


Click Copy Vector Layer
Vector Utilities
Clean Vector Layer...
Build Vector Layer Topology...
Copy Vector Layer...
External Vector Layer...
Rename Vector Layer...
Delete Vector Layer...
Display Vector Layer Info...
Subset Vector Layer...
Mosaic Polygon Layers...
Transform Vector Layer...
Create Polygon Labels...
Raster to Vector...
Vector to Raster...
Start Table Tool...
Zonal Attributes...
ASCII to Point Vector Layer...
Close
Help

2. Click the **Copy Vector Layer** button on the **Vector Utilities** menu.

The Copy Vector Layer dialog opens.


Copy Vector Layer
../
info/
lanier.dms/
bldgs
streets
va_bldgs
zone88
../
info/
examples/
data/
OK
Batch
Cancel
Help

Vector Layer to Copy: (*.arcinfo) zone88

Output Vector Layer: (*.arcinfo) zone88

Keep the name of the file the same

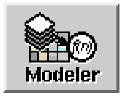
This is a directory in which you have write permission

3. In the **Vector Layer to Copy** section, navigate to the <IMAGINE_HOME>/examples directory, and select **zone88**.
4. In the **Output Vector Layer** section, navigate to a directory where you have write permission.
5. Type the name **zone88**, then press Enter on your keyboard.
6. Click **OK** in the Copy Vector Layer dialog.

   A Job Status dialog opens tracking the progress.
7. When the job is finished, click **OK** in the Job Status dialog.
8. Click **Close** in the **Vector Utilities** menu.

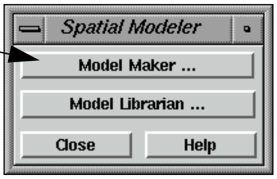
Set up the Model

1. Click the Modeler icon on the ERDAS IMAGINE icon panel.



The **Spatial Modeler** menu opens.

Select Model Maker


Spatial Modeler
Model Maker ...
Model Librarian ...
Close
Help

2. Click on the **Model Maker** button in the **Spatial Modeler** menu.

   A blank Model Maker viewer opens along with the tools.

3. Click on the Raster icon in the Model Maker tool palette.

4. Click near the upper left corner of the Model Maker viewer.

5. Click on the Vector icon in the Model Maker tool palette.

6. Click to the right of the Raster object in the Model Maker viewer.

7. Click on the Function icon in the Model Maker tool palette.

8. Click below and between the Raster and the Vector objects in the Model Maker viewer.

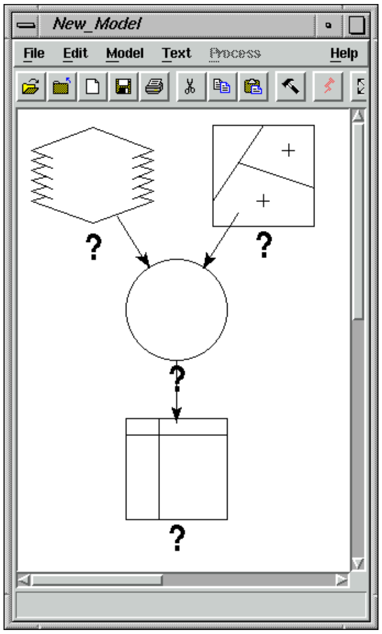
9. Click on the Table icon in the Model Maker tool palette.

10. Click below the Function object in the Model Maker viewer.

11. Using the Connection tool, and optionally the Lock tool, connect the Raster object and the Vector object to the Function object as inputs.

12. Using the Connection tool, connect the Function object to the output Table object.

    When you are finished, the model looks like the following:



Add Raster Properties

1. Confirm the Selector tool is active.

2. Double-click the Raster object.

3. In the Raster dialog, click the Open icon to open the File Name dialog.

4. Select **germtm.img** from the /examples directory and click **OK** in the File Name dialog.

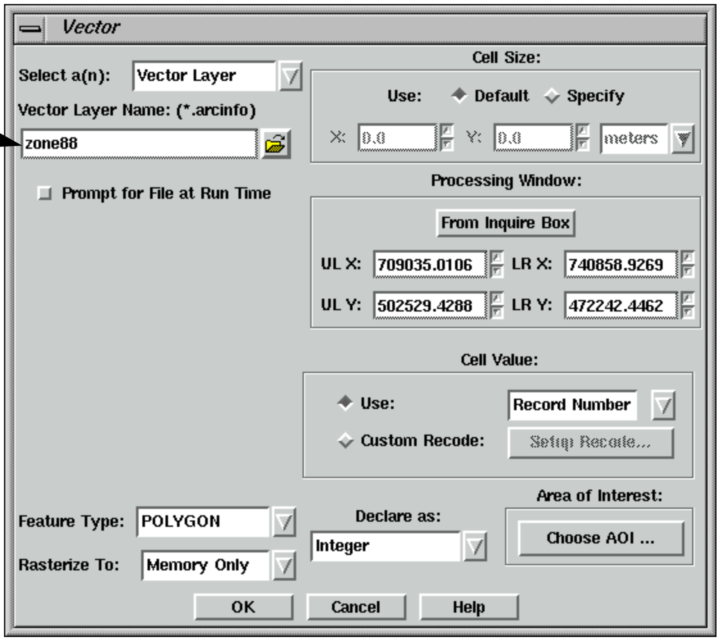
5. Click **OK** in the Raster dialog to accept the file **germtm.img**.

Add Vector Properties

1. Double-click the Vector object.

   The Vector dialog opens.

Select the file from the directory in which you copied it


Vector
Cell Size:
Processing Window:
From Inquire Box
Cell Value:
Area of Interest:
Choose AOI ...
Setup Recode...
OK
Cancel
Help

Select a(n): Vector Layer

Vector Layer Name: (*.arcinfo)

zone88

☐ Prompt for File at Run Time

Use: ● Default ○ Specify

X: 0.0 Y: 0.0 meters

UL X: 709035.0106 LR X: 740858.9269

UL Y: 502529.4288 LR Y: 472242.4462

● Use: Record Number

○ Custom Recode:

Feature Type: POLYGON

Rasterize To: Memory Only

Declare as: Integer

2. Click the Open icon to open the Vector Layer Name dialog.

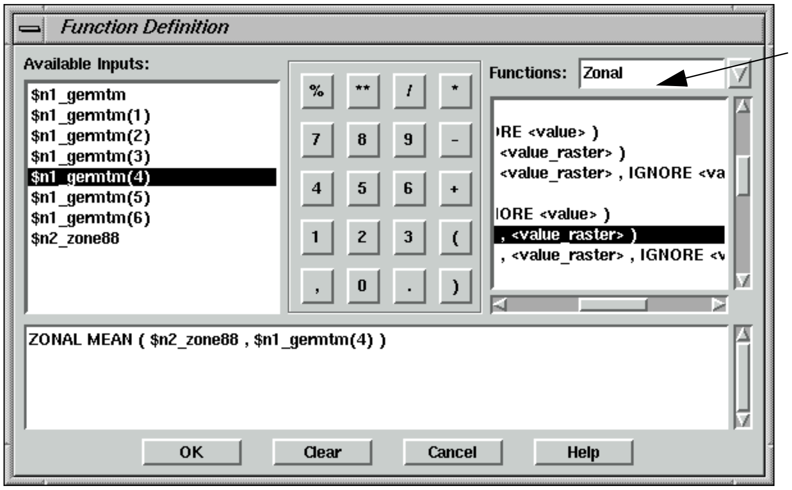
3. Select the copy of **zone88** you made earlier, and click **OK** in the Vector Layer Name dialog.

4. Click **OK** in the Vector dialog.

Add Function Properties

1. Double-click the Function object.

   The Function Definition dialog opens.


Function Definition
%
**
/
*
7
8
9
-
4
5
6
+
1
2
3
(
,
0
.
)
OK
Clear
Cancel
Help

Available Inputs:

$n1_germtm
$n1_germtm(1)
$n1_germtm(2)
$n1_germtm(3)
$n1_germtm(4)
$n1_germtm(5)
$n1_germtm(6)
$n2_zone88

Functions: Zonal

ZONAL MEAN ($n2_zone88 , $n1_germtm(4))

Choose the Zonal Function

2. In the **Functions** popup list, select **Zonal**.
3. In the list of **Functions**, select **ZONAL MEAN (<zone_raster> , <value_raster>)**, this should be the sixteenth item on the list.
4. In the lower portion of the Function Definition dialog, click in the middle of **<zone_raster>**.
5. Under **Available Inputs**, click on **$n2_zone88**.
6. In the lower portion of the Function Definition dialog, click in the middle of **<value_raster>**.
7. Under **Available Inputs**, click on **$n1_germtm(4)**.
8. Click on the **OK** button in the Function Definition dialog.

Add Table Properties

1. Double-click the Table object.

   The Table Definition dialog opens.

2. Verify that **Output** is selected.
3. Under the **Output Options**, click on the **Output to Descriptor or Attribute** checkbox.
4. For the **Existing Layer Type** popup list, select **Vector Layer**.
5. For **File**, select the copy of **zone88** from the directory in which you saved it earlier.
6. Since we are computing a new attribute, for **Attribute**, type **MEAN-NIR**.
7. The **Data Type** popup list should now be enabled, so select **Float**.

   The Table Definition dialog should look as follows:

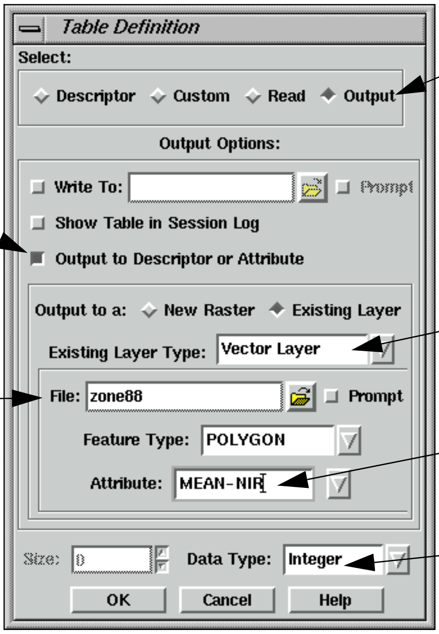


Table Definition

Select: ◇ Descriptor ◇ Custom ◇ Read ◆ Output

Output Options:

☐ Write To: ☐ Prompt

☐ Show Table in Session Log

☒ Output to Descriptor or Attribute

Output to a: ◇ New Raster ◆ Existing Layer

Existing Layer Type: Vector Layer

File: zone88 ☐ Prompt

Feature Type: POLYGON

Attribute: MEAN-NIR

Size: 0 Data Type: Integer

OK
Cancel
Help

Make sure Output is selected

Click Output to Descriptor or Attribute

Choose an existing vector layer

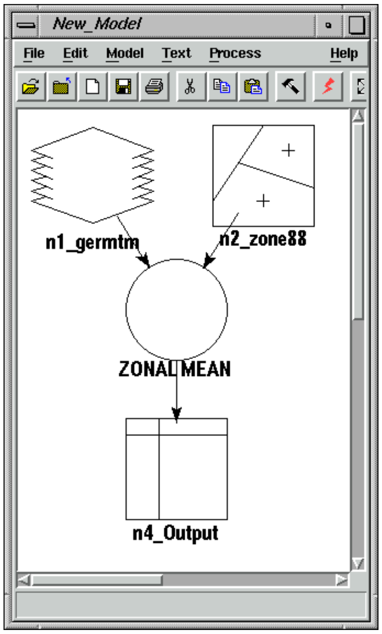
Select zone88

Name a new Attribute

Data type is integer

8. Click **OK** in the Table Definition dialog.

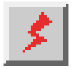
Your model should now look like the following:



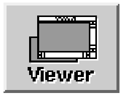
New_Model
File
Edit
Model
Text
Process
Help
n1_germtm
n2_zone88
ZONAL MEAN
n4_Output

Execute the Model and Check the Results

1. Select **Process | Run**, or click on the Execute the Model icon in the tool bar.



2. When the Job Status dialog is 100% complete, click **OK**.
3. Click the Viewer icon in the ERDAS IMAGINE icon bar.



4. In the Viewer, select **Open | Vector Layer** from the **File** menu.
5. In the Select Layer To Add dialog, select the copy of **zone88** you created and click **OK**.
6. From the Viewer’s **Vector** menu select **Attributes**.
7. In the Attribute CellArray, scroll to the right to see the newly created **MEAN-NIR** field.

The values in this column represents the mean pixel value from band 4 (near infrared) of **germtm.img** for each of the polygons in **zone88**.

Debug Your Model

Model Maker facilitates creating a model to accomplish your task, but it may still take some effort to get your model running successfully. Model Maker works hand in hand with Modeler.

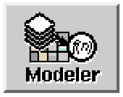
Model Maker is used to create models graphically. To execute these models, Model Maker creates an SML script, which it hands off to Modeler for execution. Modeler does all of the syntax and error checking, so finding an error in your model is not a single step operation. The following exercises demonstrate some of the common errors encountered in building new models.

Eliminate Incomplete Definition

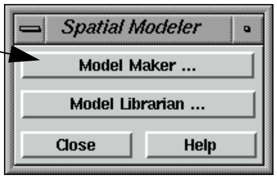
In building a model, Model Maker provides prototypes for function arguments to be replaced with actual arguments. In this exercise, you see what happens if you forget to replace a prototype.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon on the ERDAS IMAGINE icon panel.



The **Spatial Modeler** menu opens.



Select Model Maker

2. Click **Model Maker** on the **Spatial Modeler** menu.

   A blank Spatial Modeler viewer opens along with the Model Maker tool palette.

Create the Model

1. Click on the Raster icon in the Model Maker tool palette.

2. Click to place a Raster object in the upper left corner of the Model Maker viewer.

3. Click on the Matrix icon in the Model Maker tool palette.

4. Click to place the Matrix object to the right of the Raster object in the Model Maker viewer.

5. Click on the Function icon in the Model Maker tool palette.

6. Click to place the Function object below and centered between the Raster object and the Matrix object in the Model Maker viewer.

7. Click on the Raster icon in the Model Maker tool palette.

8. Click to place the Raster object below the Function object in the Model Maker viewer.

9. Click on the Connection icon in the Model Maker tool palette.

10. Click on the Lock icon in the Model Maker tool palette. It changes to reflect the locked state.

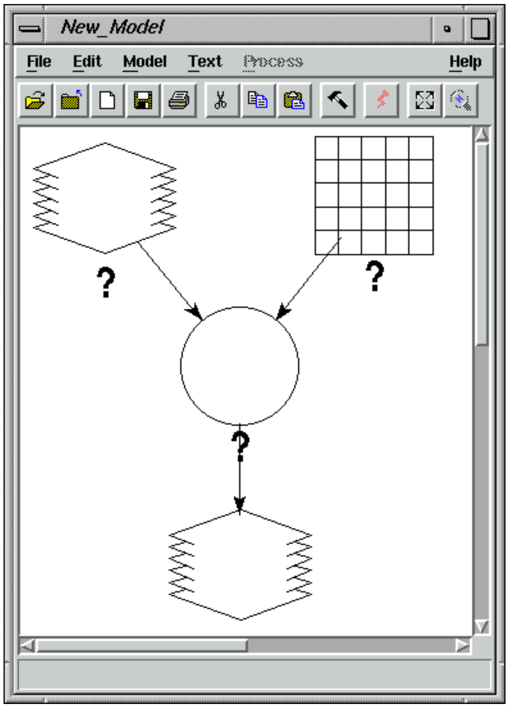
11. Connect the first Raster object and the Matrix object to the Function object as inputs.

12. Connect the Function object to the final Raster object as an output.

13. Click on the Selector icon in the Model Maker tool palette.

14. Click on the Lock icon in the Model Maker tool palette to turn it off.

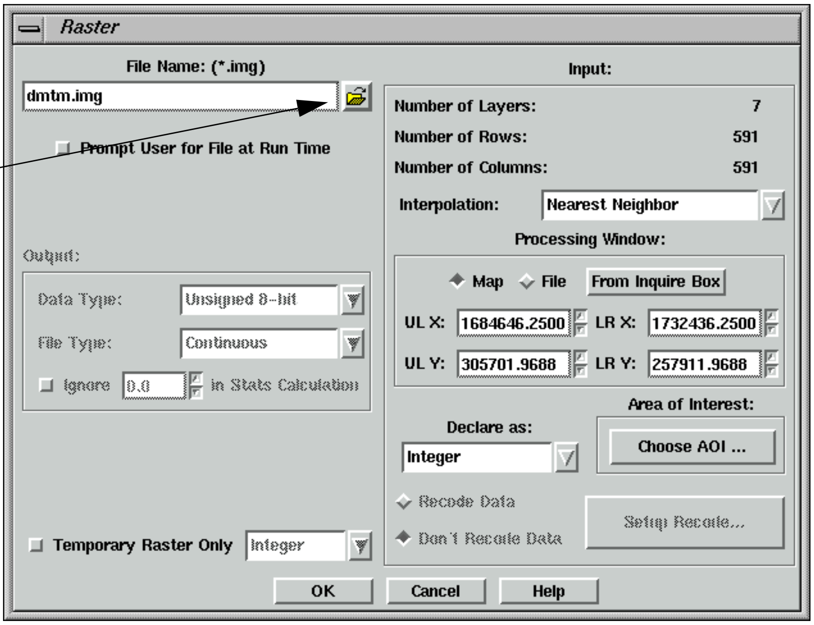
Your model now looks like the following:



Add Raster Properties

1. Double-click the first Raster object.

The Raster dialog opens.



File Name: (*.img) dmtm.img

☐ Prompt User for File at Run Time

Output:

Data Type: Unsigned 8-bit

File Type: Continuous

☐ Ignore 0.0 in Stats Calculation

☐ Temporary Raster Only Integer

Input:

Number of Layers:	7
Number of Rows:	591
Number of Columns:	591

Interpolation: Nearest Neighbor

Processing Window: ● Map ○ File

UL X: 1684646.2500 LR X: 1732436.2500

UL Y: 305701.9688 LR Y: 257911.9688

Area of Interest:

Declare as: Integer

○ Recode Data

● Don't Recode Data

Select the file here

2. Click the Open icon in the Raster dialog to open the File Name dialog.

3. In the File Name dialog, select the file **dmtm.img** from the /examples directory.

The file **dmtm.img** *is located in <IMAGINE_HOME>/examples, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.*

4. Click **OK** in the File Name dialog.

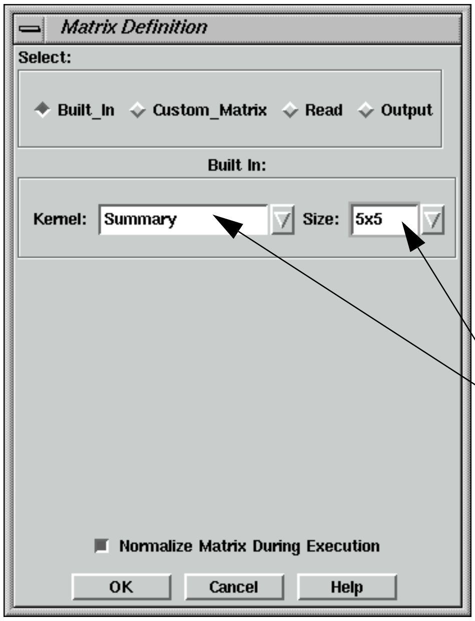
The Raster dialog updates with the appropriate **File Name**.

5. Click **OK** in the Raster dialog.

Add Matrix Properties

1. In the Model Maker viewer, double-click the Matrix object.

The Matrix Definition dialog opens.

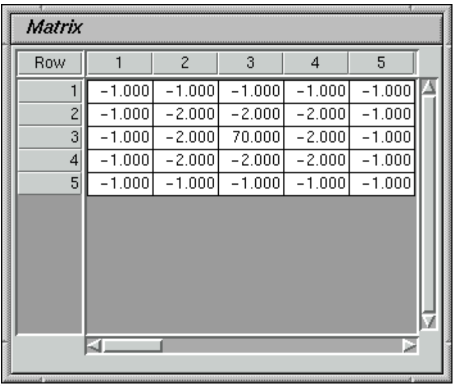


Select: ◆ Built_In ◇ Custom_Matrix ◇ Read ◇ Output

Kernel: Summary

Size: 5x5

■ Normalize Matrix During Execution



Row	1	2	3	4	5
1	-1.000	-1.000	-1.000	-1.000	-1.000
2	-1.000	-2.000	-2.000	-2.000	-1.000
3	-1.000	-2.000	70.000	-2.000	-1.000
4	-1.000	-2.000	-2.000	-2.000	-1.000
5	-1.000	-1.000	-1.000	-1.000	-1.000

Choose Summary and 5 × 5

2. From the **Kernel** list, select **Summary**.

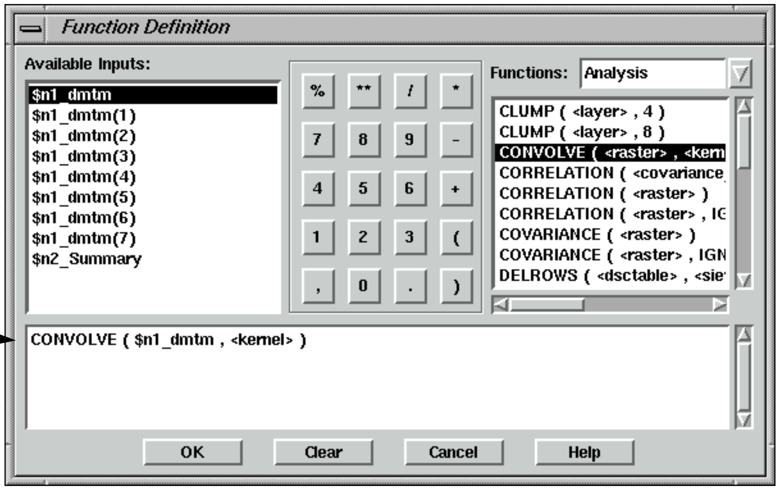
3. From the **Size** list, select **5** × **5**.

4. Click **OK** in the Matrix Definition dialog.

Add Function Properties

1. Double-click the Function object.

   The Function Definition dialog opens.


Function Definition
Available Inputs:
$n1_dmtm
$n1_dmtm(1)
$n1_dmtm(2)
$n1_dmtm(3)
$n1_dmtm(4)
$n1_dmtm(5)
$n1_dmtm(6)
$n1_dmtm(7)
$n2_Summary
%
**
/
*
7
8
9
-
4
5
6
+
1
2
3
(
,
0
.
)
CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie
OK
Clear
Cancel
Help

Functions: Analysis

```
CONVOLVE ( $n1_dmtm , <kernel> )
```

Selections you make display here

2. Confirm that the **Functions** dropdown list shows **Analysis**.

3. Under **Functions**, select **CONVOLVE (<raster> , <kernel>)**.

4. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

5. Under **Available Inputs**, click on **$n1_dmtm**.

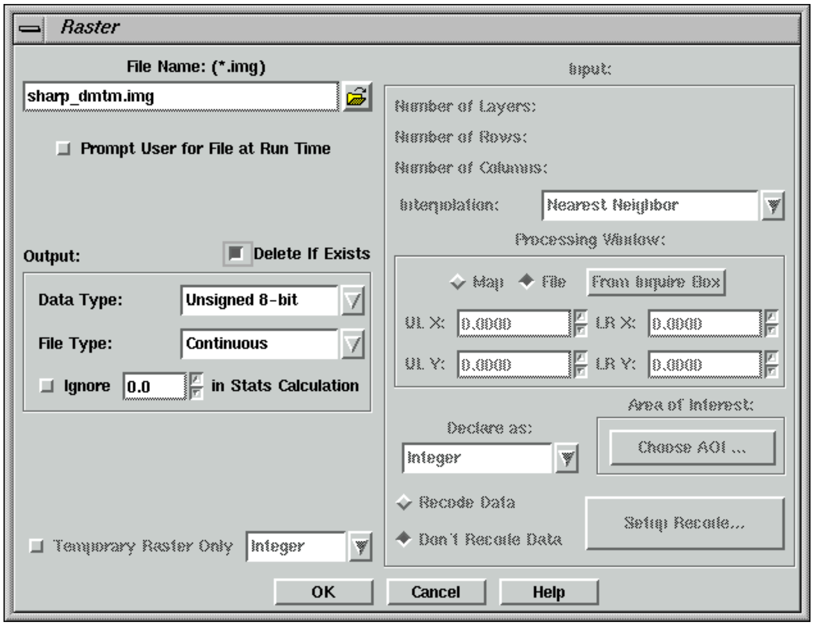
   At this point you would normally replace the **<kernel>** prototype, but you are going to intentionally forget to do that.

6. Click the **OK** button.

Add Raster Properties

1. Double-click the output Raster object.

   The Raster dialog opens.



File Name: (*.img) sharp_dmtm.img

☐ Prompt User for File at Run Time

Output: ☑ Delete If Exists

Data Type: Unsigned 8-bit

File Type: Continuous

☐ Ignore 0.0 in Stats Calculation

☐ Temporary Raster Only Integer

Input:

Number of Layers:

Number of Rows:

Number of Columns:

Interpolation: Nearest Neighbor

Processing Window:

○ Map ● File

UL X: 0.0000 LR X: 0.0000

UL Y: 0.0000 LR Y: 0.0000

Area of Interest:

Declare as: Integer

○ Recode Data

● Don't Recode Data

2. Click the Open icon, then navigate to a directory where you have write permission.

3. In the File Name dialog, enter **sharp_dmtm.img** for the output image name.

4. Click **OK** in the File Name dialog.

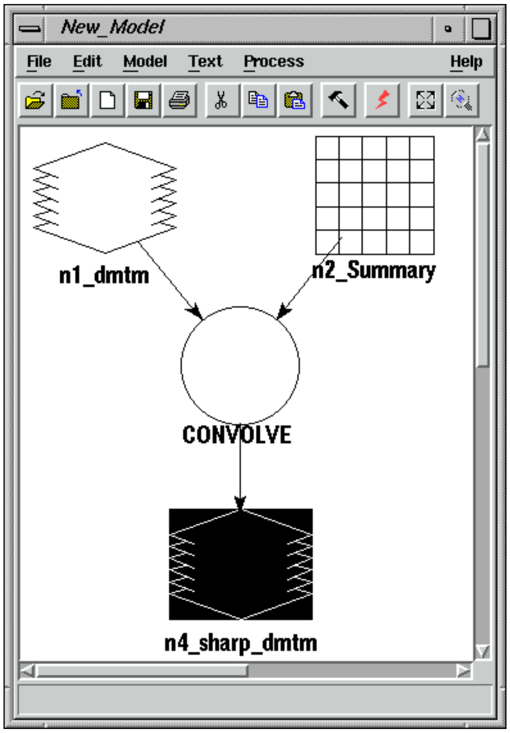
   The new file, **sharp_dmtm.img**, is listed in the Raster dialog.

5. Click on the **Delete If Exists** checkbox.

   This option allows you to run the model many times. You may have to run the model more than once to get it working.

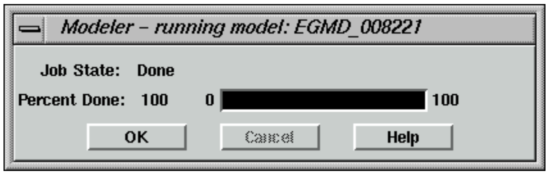
6. Click on the **OK** button in the Raster dialog.

   At this point, your model should look similar to the following:


New_Model
File
Edit
Model
Text
Process
Help
n1_dmtm
n2_Summary
CONVOLVE
n4_sharp_dmtm

7. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

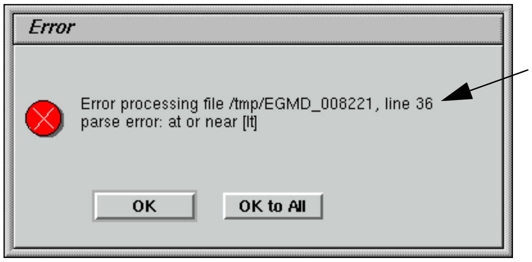
A Job Status dialog opens, which tracks the progress of the model execution.


Modeler – running model: EGMD_008221
0
100
OK
Cancel
Help

Job State: Done

Percent Done: 100

You receive an error similar to the following:



Error processing file /tmp/EGMD_008221, line 36
parse error: at or near [lt]

Note the line number the error occurs on

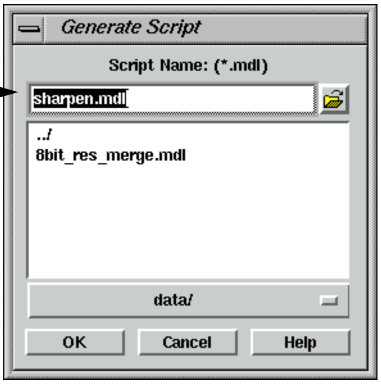
Correct the Model

The next step is to figure out what this error means.

1. Click **OK** to dismiss the Error dialog.
2. Click **OK** to dismiss the Job Status dialog.

You can set the **Keep Job Status Box** *in the* **User Interface & Session** *category of the Preference Editor so that the Job Status box is dismissed automatically after an operation is performed.*

3. In the Model Maker viewer, select **Process/Generate Script**.



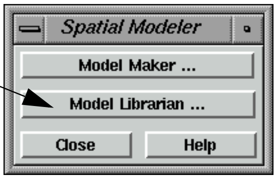
Script Name: (*.mdl)
sharpen.mdl

The name of the model displays here

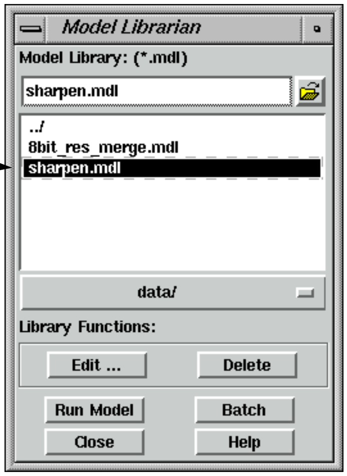
4. In the Generate Script dialog, click the Open icon, and navigate to a directory where you have write permission.
5. Enter the name **sharpen.mdl**, and click **OK**.

Start the Model Librarian

1. In the Spatial Modeler dialog, click on the **Model Librarian** button.

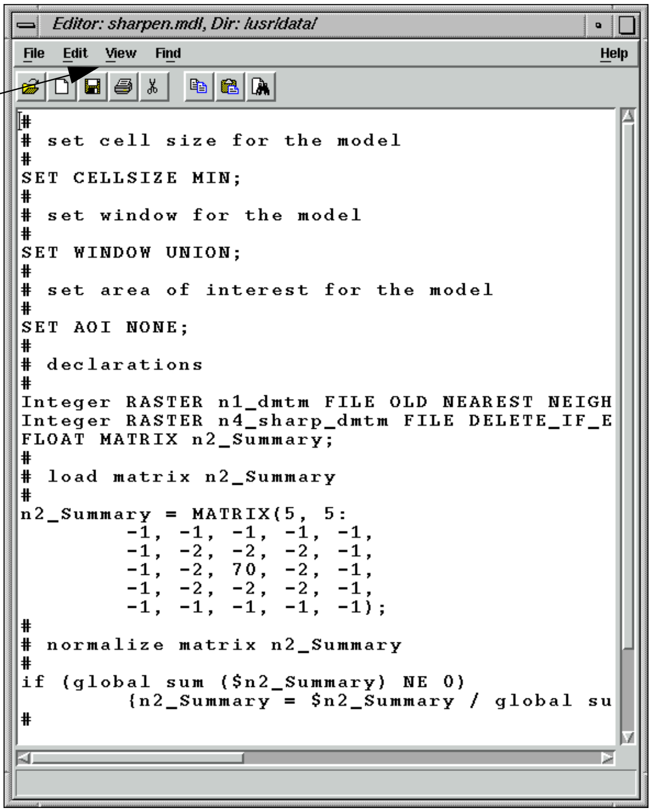

Select Model Librarian
Spatial Modeler
Model Maker ...
Model Librarian ...
Close
Help

The Model Librarian dialog opens.


Select the model sharpen.mdl from the list
Model Librarian
Model Library: (*.mdl)
sharpen.mdl
../
8bit_res_merge.mdl
sharpen.mdl
data/
Library Functions:
Edit ...
Delete
Run Model
Batch
Close
Help

2. Navigate to the correct directory, then select **sharpen.mdl**.

3. Click on the **Edit** button in the Model Librarian dialog.

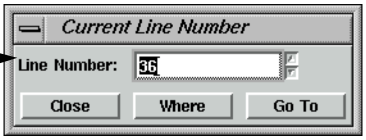


Select the Current Line Number option from this menu

```
#
# set cell size for the model
#
SET CELLSIZE MIN;
#
# set window for the model
#
SET WINDOW UNION;
#
# set area of interest for the model
#
SET AOI NONE;
#
# declarations
#
Integer RASTER n1_dmtm FILE OLD NEAREST NEIGH
Integer RASTER n4_sharp_dmtm FILE DELETE_IF_E
FLOAT MATRIX n2_Summary;
#
# load matrix n2_Summary
#
n2_Summary = MATRIX(5, 5:
        -1, -1, -1, -1, -1,
        -1, -2, -2, -2, -1,
        -1, -2, 70, -2, -1,
        -1, -2, -2, -2, -1,
        -1, -1, -1, -1, -1);
#
# normalize matrix n2_Summary
#
if (global sum ($n2_Summary) NE 0)
        {n2_Summary = $n2_Summary / global su
#
```

4. In the Editor window, select **View | Current Line Number**.

   The Current Line Number dialog opens.



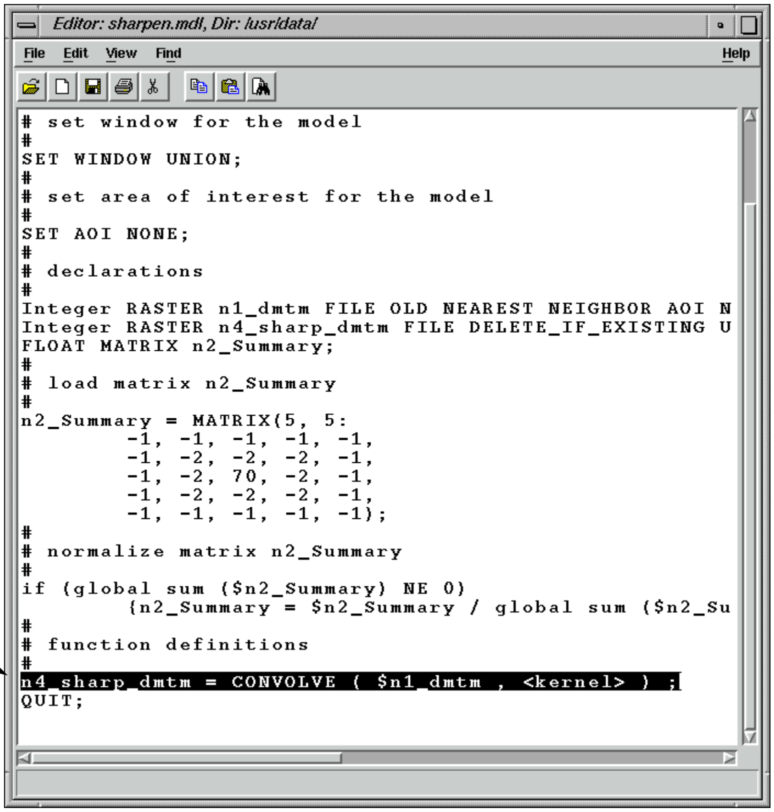
Line Number: 36

Enter the Line Number you want to view

5. In the Current Line Number dialog, enter **36** for the **Line Number** (the line number referred to in the Error dialog).

6. Click the **Go To** button.

   This highlights the line containing the error as depicted in the following picture:



```
# set window for the model
#
SET WINDOW UNION;
#
# set area of interest for the model
#
SET AOI NONE;
#
# declarations
#
Integer RASTER n1_dmtm FILE OLD NEAREST NEIGHBOR AOI N
Integer RASTER n4_sharp_dmtm FILE DELETE_IF_EXISTING U
FLOAT MATRIX n2_Summary;
#
# load matrix n2_Summary
#
n2_Summary = MATRIX(5, 5:
        -1, -1, -1, -1, -1,
        -1, -2, -2, -2, -1,
        -1, -2, 70, -2, -1,
        -1, -2, -2, -2, -1,
        -1, -1, -1, -1, -1);
#
# normalize matrix n2_Summary
#
if (global sum ($n2_Summary) NE 0)
        {n2_Summary = $n2_Summary / global sum ($n2_Su
#
# function definitions
#
n4_sharp_dmtm = CONVOLVE ( $n1_dmtm , <kernel> ) ;
QUIT;
```

The line, 36, is highlighted

If you examine the selected line, just to the right of the equal sign is a function, which also serves as a label to a Function object in the graphical model. Most syntax errors occur in Function definitions. In general, you generate a script so you can relate the line number given in the error message back to a particular Function object in the model.

Correct the Function

1. In the Model Maker viewer, double-click the Function object, **CONVOLVE**.

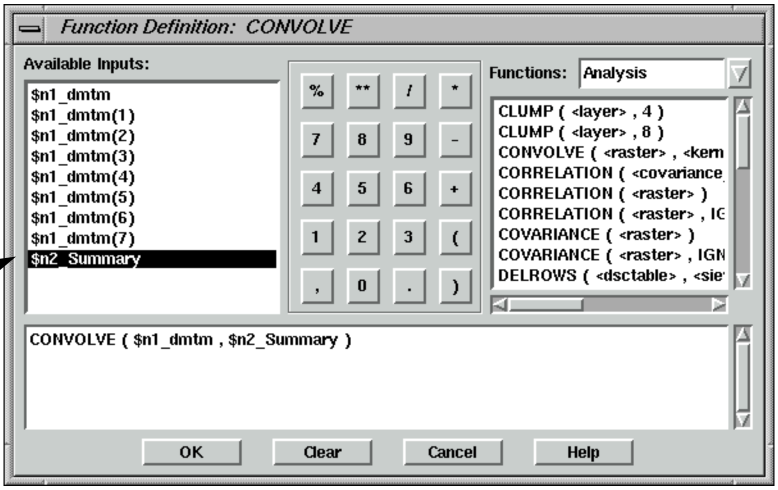
2. Examine the function definition to determine the error.

   In this case, you determine the function definition still has an argument prototype, **<kernel>**, that needs to be replaced with an actual argument.

3. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.

4. Under **Available Inputs**, click on **$n2_Summary**.

   Your Function Definition dialog now looks like the following:



Available Inputs:

$n1_dmtm
$n1_dmtm(1)
$n1_dmtm(2)
$n1_dmtm(3)
$n1_dmtm(4)
$n1_dmtm(5)
$n1_dmtm(6)
$n1_dmtm(7)
$n2_Summary

Functions: Analysis

CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie

CONVOLVE ($n1_dmtm , $n2_Summary)

The input $n2_Summary replaces <kernel>

5. Click **OK** in the Function Definition dialog.

Execute the Model

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

   The model should run to completion without error this time.

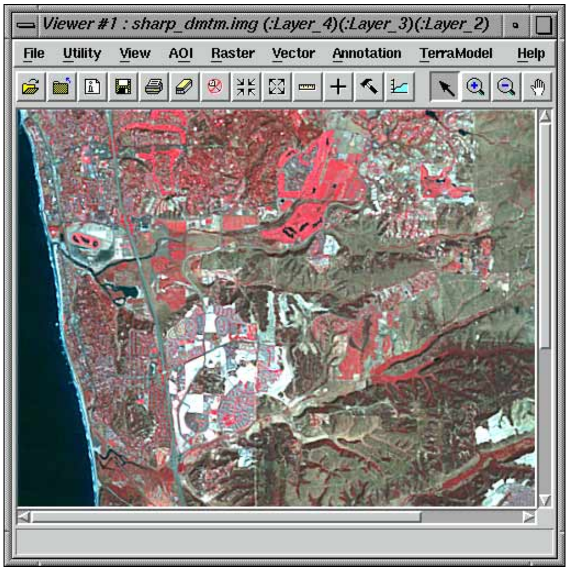
2. Click on the **OK** button to dismiss the Job Status dialog.

Check the Results

1. Click the Viewer icon to open a new Viewer.

2. Click the Open icon, then navigate to the directory where you saved the file **sharp_dmtm.img**.

3. Select the file **sharp_dmtm.img**, then click **OK** in the Select Layer To Add dialog to open it in the Viewer.



4. When you are finished viewing the image, click **File | Close** in the Viewer to close the image.

Eliminate Object type Mismatch

There are five basic object types that can be either inputs to or outputs from a model. These are:

- Raster
- Vector (input only)
- Matrix
- Table
- Scalar

Depending on the arguments, each function produces a particular object type.

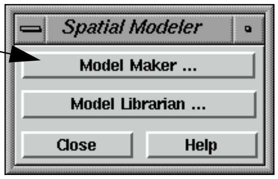
For example, the GLOBAL MAX function produces a Scalar if the argument is either a Matrix or a Table. However, it produces a Table if the argument is a Raster. In other words, for either a Matrix or a Table, the maximum value may be represented by a single number (i.e., Scalar). A Raster has a maximum value in each individual spectral band, so the result in this case is a Table of maximum values: one for each band. In order to be consistent, this is still true for a Raster with only one band. In this case a table is produced with a single entry.

In the following exercise, you build a model that rescales an image based on the maximum pixel value that actually occurs in an image. You do this using the GLOBAL MAX function. Initially, you incorrectly treat the output of the GLOBAL MAX function as a Scalar so you can see the type of error generated.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon on the ERDAS IMAGINE icon panel.

The **Spatial Modeler** menu opens.



Select Model Maker

2. Click **Model Maker** on the **Spatial Modeler** menu.

   A blank Spatial Modeler viewer opens along with the Model Maker tool palette.

Create the Model

1. Click on the Raster icon in the Model Maker tool palette.
2. Click to position the Raster object in the upper left corner of the Model Maker viewer.
3. Click on the Function icon in the Model Maker tool palette.
4. Click to position the Function object below and to the right of the Raster object in the Model Maker viewer.
5. Click on the Scalar icon in the Model Maker tool palette.
6. Click to position the Scalar object below and to the right of the Function object in the Model Maker viewer.
7. Click on the Function icon in the Model Maker tool palette.
8. Click to position the Function object to the left of the Scalar object in the Model Maker viewer.
9. Click on the Raster icon in the Model Maker tool palette.

10. Click to position the Raster object below the Scalar object in the Model Maker viewer.

11. Click on the Connection icon in the Model Maker tool palette.

12. Click on the Lock icon in the Model Maker tool palette. It changes to the locked state.

13. Connect the first Raster to the first Function.

14. Also connect the first Raster to the second Function.

15. Connect the first Function to the Scalar.

16. Connect the Scalar to the second Function.

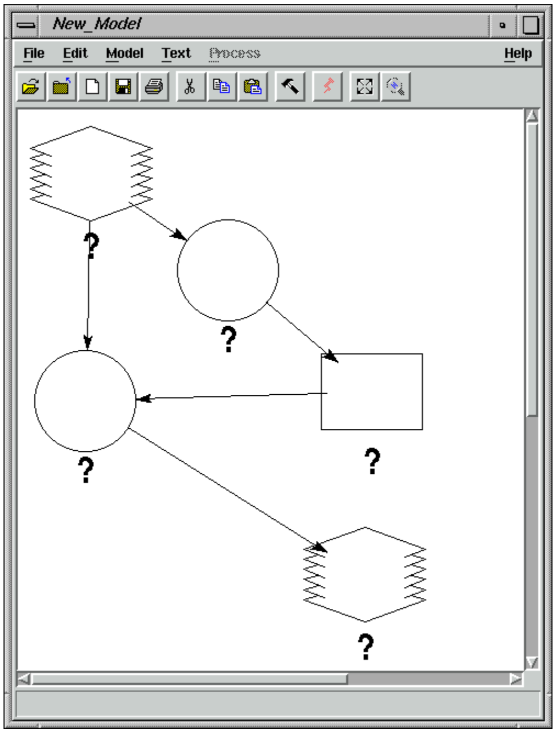
17. Connect the second Function to the final output Raster.

NOTE: You may want to refer to the following diagram of the model to verify your connections. Connections may be broken or deleted by using the Connection tool in the reverse direction of the existing connection.

18. Click on the Selector icon in the Model Maker tool palette.

19. Click on the Lock icon in the Model Maker tool palette to turn it off.

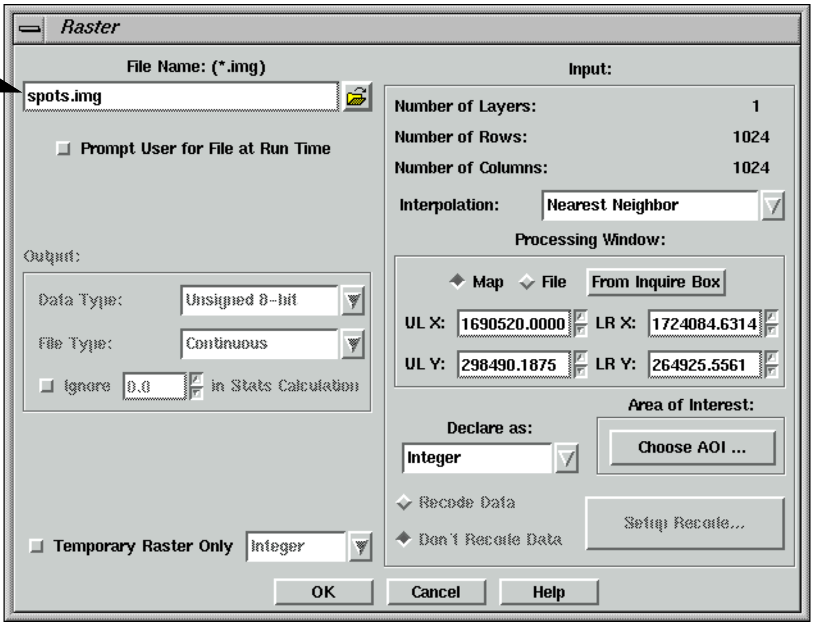
Your model should look like the following:


New_Model
File
Edit
Model
Text
Process
Help
?
?
?
?
?

Add Raster Properties

1. Double-click the first Raster object.

   The Raster dialog opens.



Select the file spots.img

Raster

File Name: (*.img) spots.img

☐ Prompt User for File at Run Time

Output:

Data Type: Unsigned 8-bit

File Type: Continuous

☐ Ignore 0.0 in Stats Calculation

☐ Temporary Raster Only Integer

Input:

Number of Layers: 1

Number of Rows: 1024

Number of Columns: 1024

Interpolation: Nearest Neighbor

Processing Window:

● Map ○ File From Inquire Box

UL X: 1690520.0000 LR X: 1724084.6314

UL Y: 298490.1875 LR Y: 264925.5561

Declare as: Integer

Area of Interest: Choose AOI ...

○ Recode Data

● Don't Recode Data

Setup Recode...

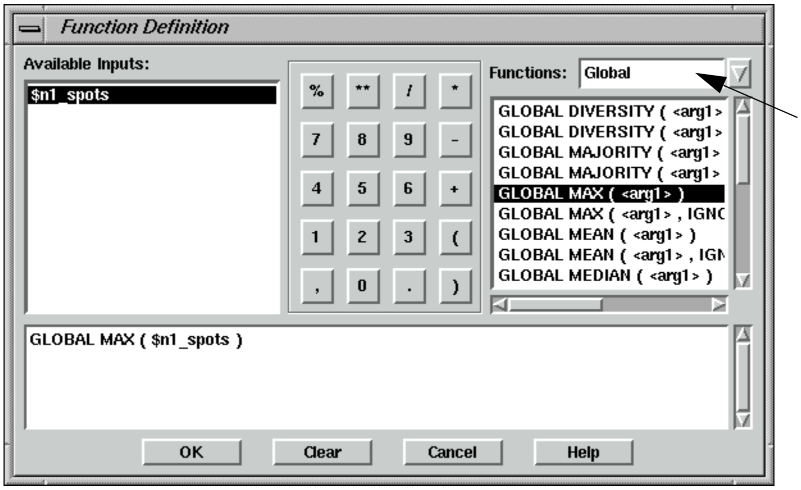
2. Click the Open icon on the Raster dialog, and navigate to the /examples directory.

3. In the Open File dialog, select **spots.img** and click **OK**.

Add Function Properties

1. Double-click the first Function object.

   The Function Definition dialog opens.


Function Definition
%
**
/
*
7
8
9
-
4
5
6
+
1
2
3
(
,
0
.
)
GLOBAL DIVERSITY (<arg1>
GLOBAL DIVERSITY (<arg1>
GLOBAL MAJORITY (<arg1>
GLOBAL MAJORITY (<arg1>
GLOBAL MAX (<arg1>)
GLOBAL MAX (<arg1> , IGNC
GLOBAL MEAN (<arg1>)
GLOBAL MEAN (<arg1> , IGN
GLOBAL MEDIAN (<arg1>)
OK
Clear
Cancel
Help

Available Inputs: $n1_spots

Functions: Global

GLOBAL MAX ($n1_spots)

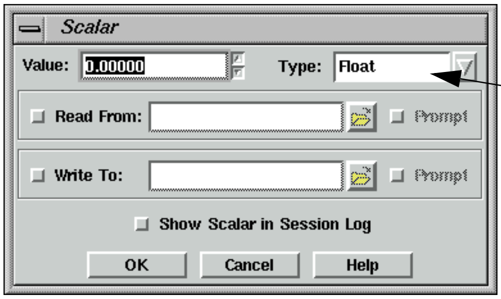
Select Global from the Functions list

2. In the **Functions** popup list, select **Global**.
3. In the list of **Global** functions, select **GLOBAL MAX (<arg1>)**.
4. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.
5. Under **Available Inputs**, click on **$n1_spots**.
6. Click **OK** in the Function Definition dialog.

Add Scalar Properties

1. Double-click the Scalar object.

   The Scalar dialog opens.


Scalar
OK
Cancel
Help

Value: 0.00000

Type: Float

☐ Read From: ☐ Prompt

☐ Write To: ☐ Prompt

☐ Show Scalar in Session Log

Confirm that the Type is set to Float

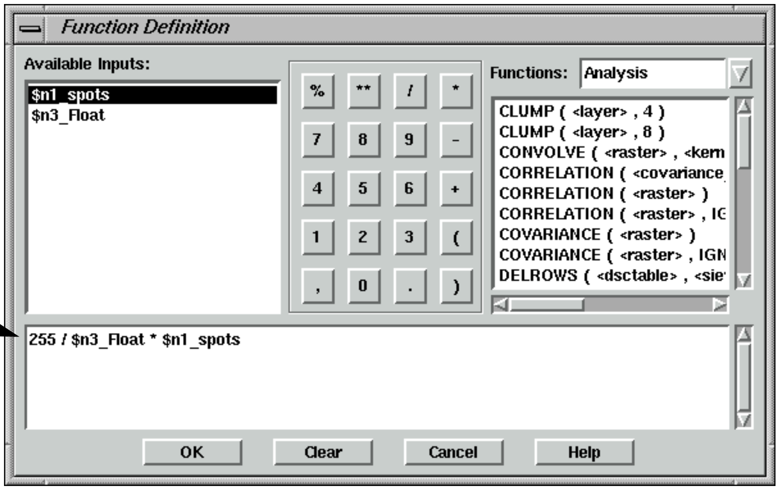
2. Verify that the **Type** is set to **Float**, and click **OK**.

You select **Float** to insure the model uses floating point arithmetic instead of integer arithmetic. You do this because you are calculating a ratio between 255 and the GLOBAL MAX. In other words, you want to be able to multiply the pixel values by numbers such as 1.3, 2.1, or 3.4 and not just 1, 2, or 3.

Add Function Properties

1. Double-click the second Function object.

   The Function dialog opens.


Function Definition
Available Inputs:
$n1_spots
$n3_Float
% ** / *
7 8 9 -
4 5 6 +
1 2 3 (
, 0 .)
Functions:
CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie
OK
Clear
Cancel
Help

Functions: Analysis

255 / $n3_Float * $n1_spots

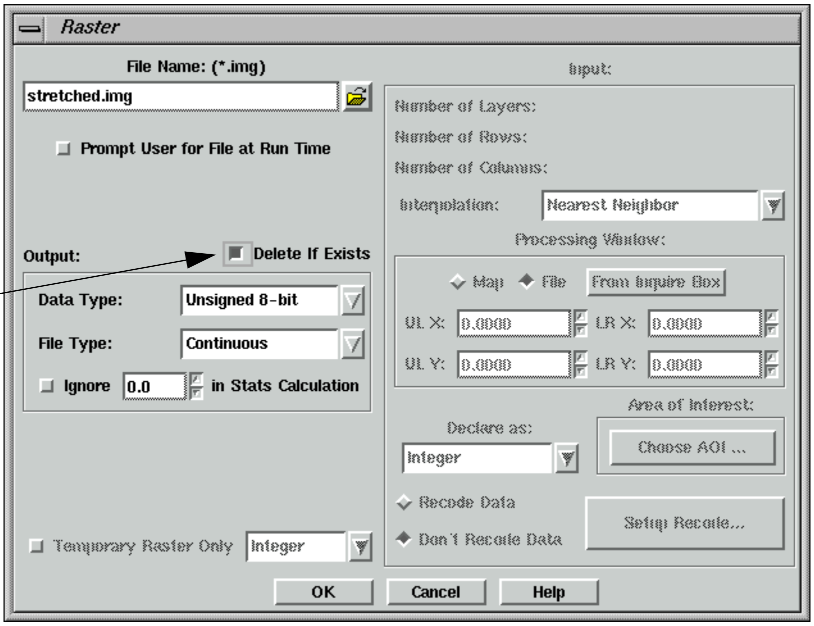
The function definition appears here

2. Using the calculator portion of the Function Definition dialog enter **255 /** .
3. Under **Available Inputs** click on **$n3_Float**.
4. In the calculator portion of the Function Definition dialog click on *****.
5. Under **Available Inputs** click on **$n1_spots**.
6. Click **OK** in the Function Definition dialog.

Add Raster Properties

1. Double-click the output Raster object.

   The Raster dialog opens.



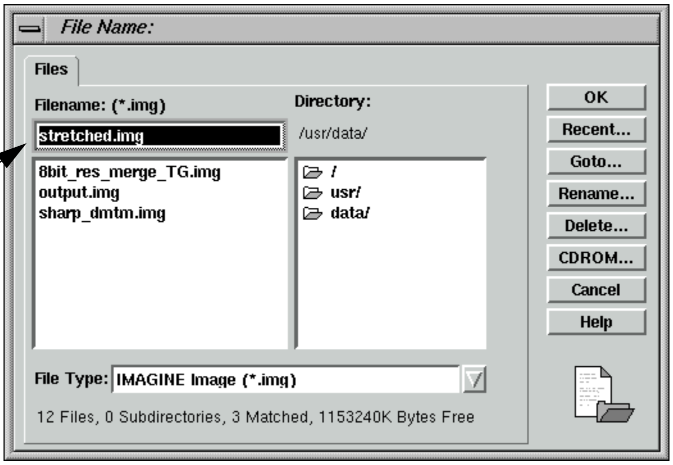
Make sure you click the Delete If Exists checkbox

File Name: (*.img) stretched.img
☐ Prompt User for File at Run Time
Output: ☒ Delete If Exists
Data Type: Unsigned 8-bit
File Type: Continuous
☐ Ignore 0.0 in Stats Calculation
☐ Temporary Raster Only Integer
Input:
Number of Layers:
Number of Rows:
Number of Columns:
Interpolation: Nearest Neighbor
Processing Window:
◇ Map ◆ File
UL X: 0.0000 LR X: 0.0000
UL Y: 0.0000 LR Y: 0.0000
Area of Interest:
Declare as: Integer
◇ Recode Data
◆ Don't Recode Data

Raster
From Inquire Box
Choose AOI ...
Setup Recode...
OK
Cancel
Help

2. Click the Open icon in the Raster dialog, then navigate to a directory where you have write permission.

3. Enter **stretched.img** for the output image name, then click **OK** in the File Name dialog.



Save the file in a directory where you have write permission

Filename: (*.img) stretched.img
Directory: /usr/data/
8bit_res_merge_TG.img
output.img
sharp_dmtm.img
/
usr/
data/
File Type: IMAGINE Image (*.img)
12 Files, 0 Subdirectories, 3 Matched, 1153240K Bytes Free

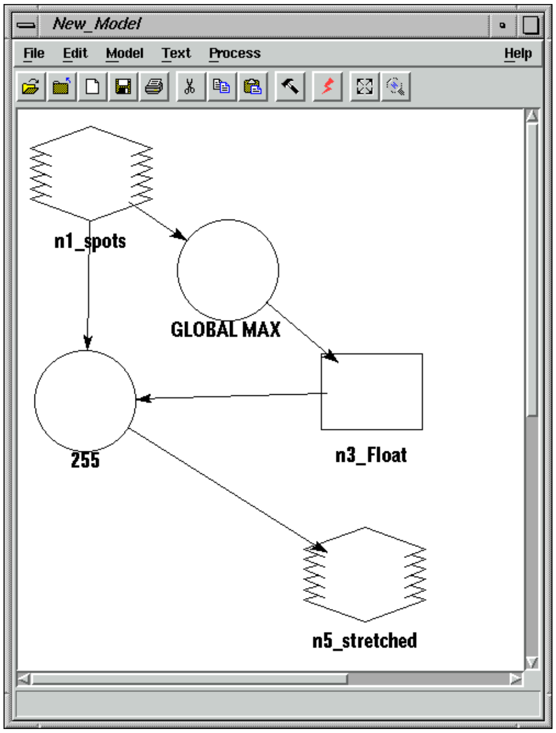
File Name:
Files
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

4. In the Raster dialog, click the **Delete If Exists** checkbox.

You may have to run the model more than once to get it working.

5. Click on the **OK** button.

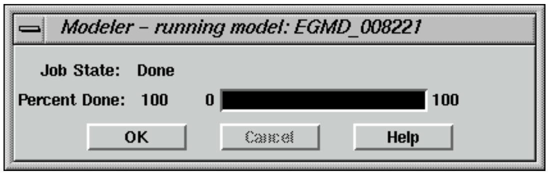
At this point, your model should look similar to the following:


New_Model
File
Edit
Model
Text
Process
Help
n1_spots
GLOBAL MAX
255
n3_Float
n5_stretched

Execute the Model

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

A Job Status dialog opens.



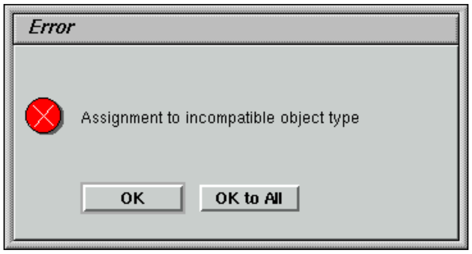
Modeler – running model: EGMD_008221

Job State: Done

Percent Done: 100

0
100
OK
Cancel
Help

You receive an error like the following:



Error

Assignment to incompatible object type

The next step is to figure out what this error means.

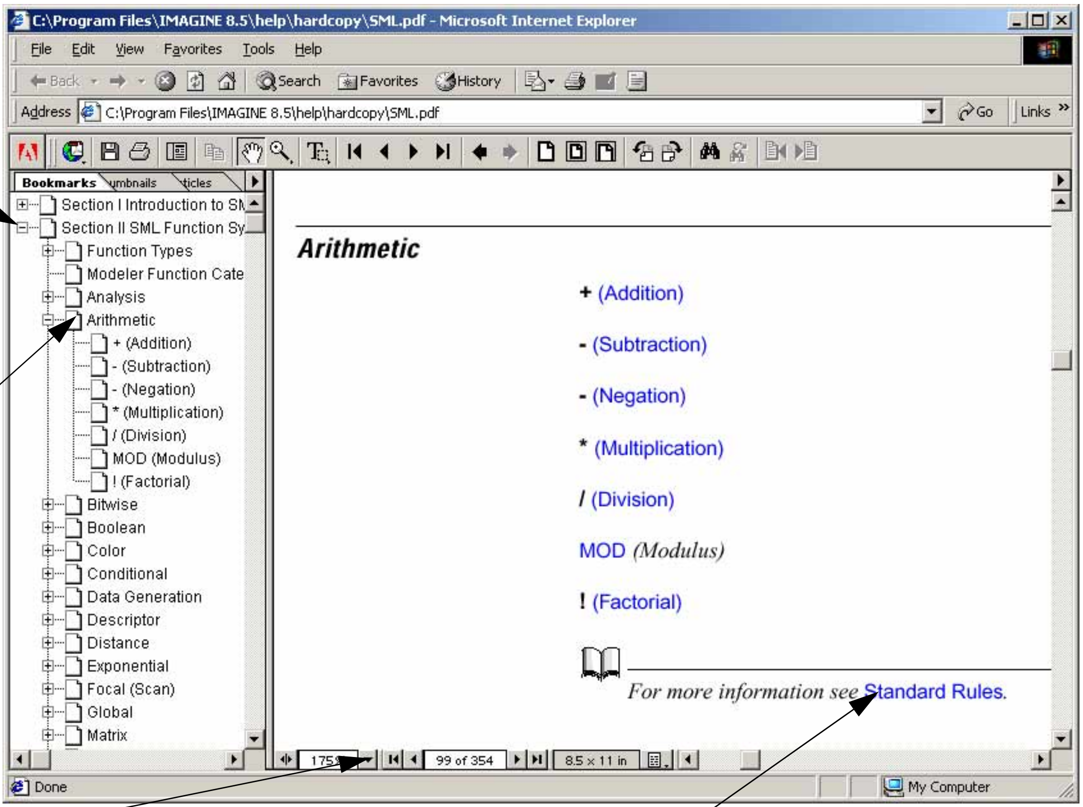
2. Click **OK** to dismiss the Error dialog.
3. Click **OK** to dismiss the Job Status dialog.

Check the On-Line Help

When a model is executed, an Assignment statement is generated for each Function object in the model. The error is telling you that one of the Function objects in the model is generating a different object type than what you have it connected to.

You know that in one of our Function objects we using the GLOBAL MAX function, and in the other you are just doing arithmetic. At this point, you can use the online documentation to help out.

1. Click **Help | Help for Model Maker**.
2. Click on the hyperlink to the **Spatial Modeler Language Reference Manual** in the third paragraph. The on-line manual is in Adobe® portable document format (SML.pdf). It will open in a new browser window.
3. In the Navigation Pane, click on the **+** beside **Section II SML Function Syntax** to view all topics included in that section.
4. Click on the **Arithmetic** topic to open that page in the Acrobat viewer.


C:\Program Files\IMAGINE 8.5\help\hardcopy\SML.pdf - Microsoft Internet Explorer
File Edit View Favorites Tools Help
Back Search Favorites History
Address C:\Program Files\IMAGINE 8.5\help\hardcopy\SML.pdf
Go Links
Bookmarks Thumbnails Articles
Section I Introduction to SM
Section II SML Function Sy
Function Types
Modeler Function Cate
Analysis
Arithmetic
+ (Addition)
- (Subtraction)
- (Negation)
* (Multiplication)
/ (Division)
MOD (Modulus)
! (Factorial)
Bitwise
Boolean
Color
Conditional
Data Generation
Descriptor
Distance
Exponential
Focal (Scan)
Global
Matrix
175%
99 of 354
8.5 x 11 in
Done
My Computer

Arithmetic

+ (Addition)

- (Subtraction)

- (Negation)

* (Multiplication)

/ (Division)

MOD *(Modulus)*

! (Factorial)

For more information see Standard Rules.

Click here to expand Section II

Click here to display the Arithmetic page

You can use the Zoom feature to adjust the zoom to a comfortable reading level

Click here to link to the Standard Rules page

5. After reading the topic, click on the hyperlink to open the **Standard Rules** page. The **Standard Rules for Combining Different Types** topic displays.

6. Scroll down to the **Object Types** section.

   While this section contains some very useful information, it gives no indication that anything is wrong with the Function in which you are doing simple arithmetic.

7. In the Navigation Pane, click on the **+** beside **Global** to view all the pages under this topic..

8. Click on **GLOBAL MAX (Global Maximum)** to display this topic.

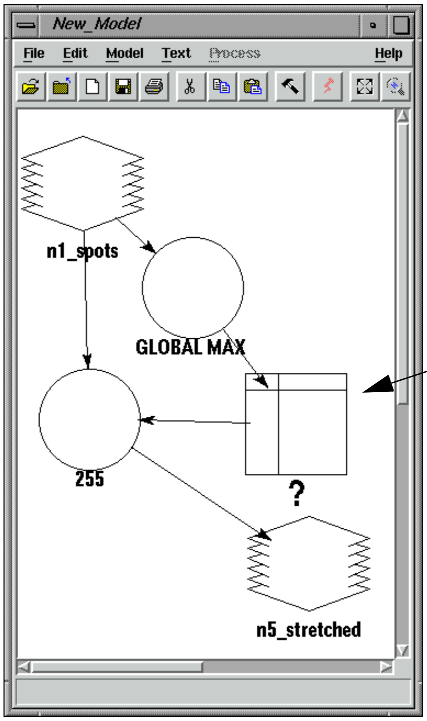
9. Scroll down to the **Object Types** section.

   Note the on-line documentation states, “If <arg1> is a RASTER, the result is a TABLE with the same number of rows as <arg1> has layers”. In your model, you incorrectly connected the output of GLOBAL MAX of a Raster to a Scalar instead of a Table.

10. Select **File | Exit** from the On-Line Help dialog.

Correct the Model

1. In the Model Maker viewer, click on the Scalar object.
2. Select **Edit | Clear**, or press the Delete key on your keyboard.
3. Click on the Table icon in the Model Maker tool palette.
4. Click to position the Table object in the location where the Scalar object was in the Model Maker viewer.
5. Using the Connection tool, connect the first Function to the Table, and the Table to the second Function.


New_Model
File
Edit
Model
Text
Process
Help
n1_spots
GLOBAL MAX
255
?
n5_stretched

The Table replaces the Scalar you originally placed in the model

Add Table Properties

1. Using the Selector tool, double-click the Table object in the Model Maker viewer.

   The Table Definition dialog opens.

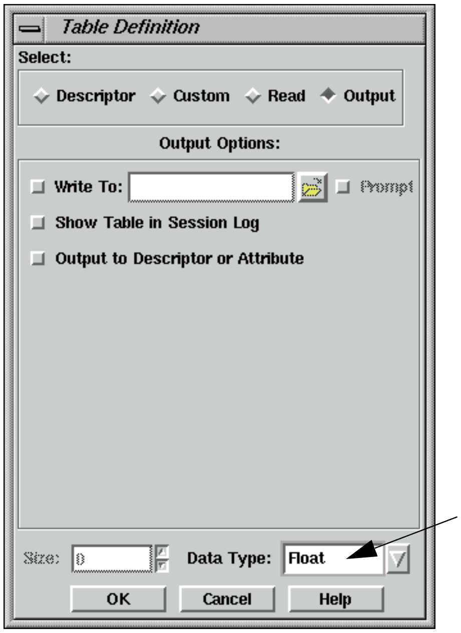


Table Definition

Select:

◇ Descriptor ◇ Custom ◇ Read ◆ Output

Output Options:

☐ Write To: ☐ Prompt

☐ Show Table in Session Log

☐ Output to Descriptor or Attribute

Size: 0 Data Type: Float

Select the Float Data Type

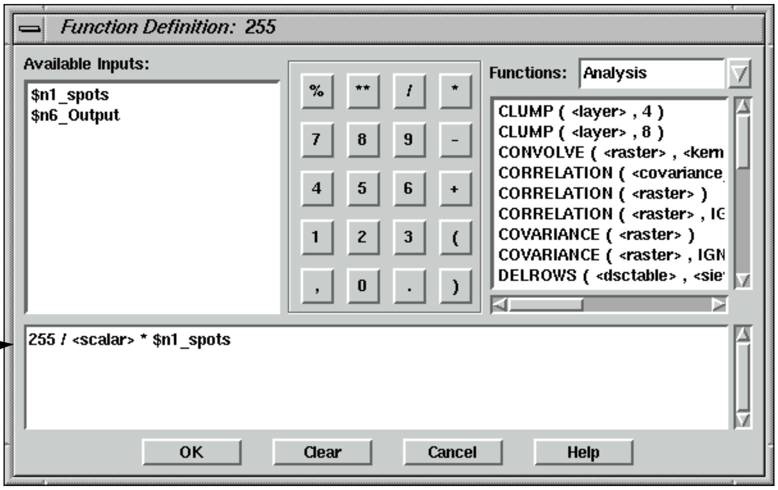
2. Click the **Data Type** popup list, and select **Float**.

3. Click on the **OK** button.

Correct Function Properties

1. Double-click the second Function object.

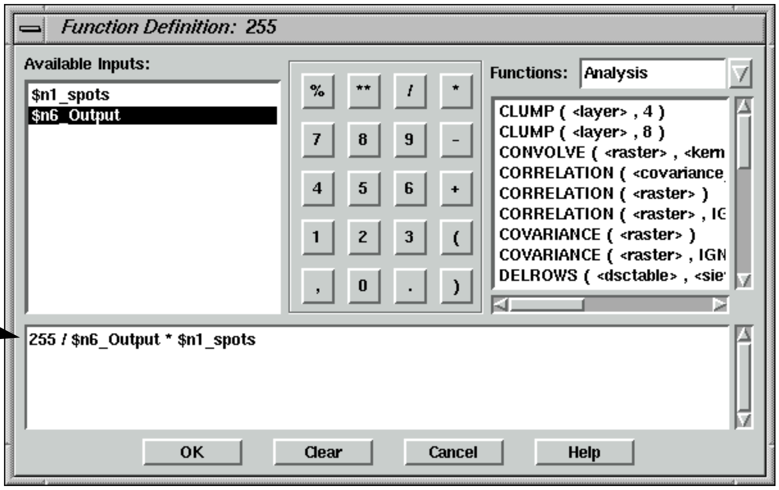
Notice Model Maker replaced the name of the deleted Scalar object with a **<scalar>** place holder. It did this to remind you what was there before. In this case, you replace **<scalar>** with a Table.


Function Definition: 255
Available Inputs:
$n1_spots
$n6_Output
%
**
/
*
7
8
9
-
4
5
6
+
1
2
3
(
,
0
.
)
Functions:
Analysis
CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie
OK
Clear
Cancel
Help

255 / <scalar> * $n1_spots

Replace the <scalar> place holder

2. In the lower portion of the Function Definition dialog, click in the middle of **<scalar>**.
3. Under **Available Inputs**, click on **$n6_Output**.


Function Definition: 255
Available Inputs:
$n1_spots
$n6_Output
%
**
/
*
7
8
9
-
4
5
6
+
1
2
3
(
,
0
.
)
Functions:
Analysis
CLUMP (<layer> , 4)
CLUMP (<layer> , 8)
CONVOLVE (<raster> , <kern
CORRELATION (<covariance
CORRELATION (<raster>)
CORRELATION (<raster> , IG
COVARIANCE (<raster>)
COVARIANCE (<raster> , IGN
DELROWS (<dsctable> , <sie
OK
Clear
Cancel
Help

255 / $n6_Output * $n1_spots

The correct input, $n6_Output displays in the window

4. Click on the **OK** button in Function Definition dialog.

Execute the Model

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

The model should run to completion without error this time.

2. Click on the **OK** button to dismiss the Job Status dialog.

   The other advantage our model has, by properly treating the output of the GLOBAL MAX function as a Table, is that it works whether the input image has only a single band or hundreds of bands. Remember that, with multispectral data, the Table generated by the GLOBAL MAX function has an entry for each band representing the maximum value in each band, respectively. When we multiply a Raster by a Table, each band in the Raster is multiplied by the corresponding entry in the Table. This allows our model to work on all bands at once without having to loop through each band.

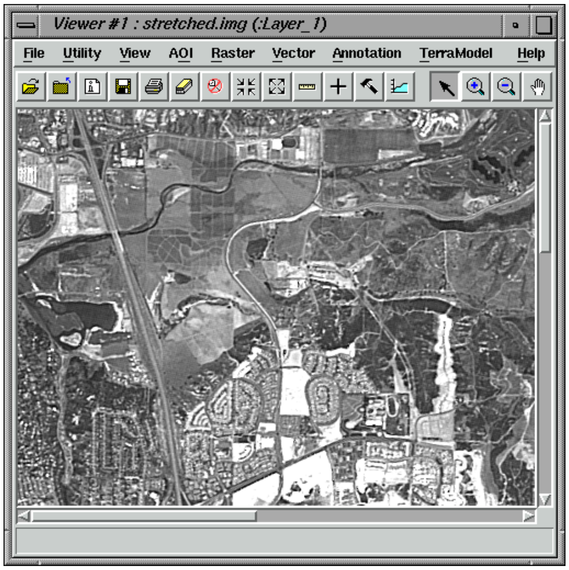
View the Results

1. Click the Viewer icon to open a new Viewer.

   Viewer

2. Click the Open icon, then navigate to the directory where you saved the file **sharp_dmtm.img**.

3. Select the file **stretched.img**, then click **OK** in the Select Layer To Add dialog to open it in the Viewer.



When you are finished, click **File | Close** in the Viewer to close the image.

Eliminate Division by Zero

Now that you are familiar with the tools and interface of the Spatial Modeler, the following two examples,"Eliminate Division by Zero" *and* "Use AOIs in Processing" *do not have detailed instructions, or as many screen captures to guide you through the process.*

Calculating band ratios with multispectral imagery is a very common image processing technique. Calculating a band ratio can be as simple as dividing one spectral band by another. Any time division is done, care should be taken to avoid division by zero, which is undefined. In the following model, you can see what errors division by zero can cause and how to correct these errors.

Create the Model

You must have Model Maker running.

1. Click on the Raster icon in the Model Maker tool palette.
2. Click near the upper left corner of the Model Maker viewer.
3. Click on the Function icon in the Model Maker tool palette.
4. Click below and to the right of the Raster object in the Model Maker viewer.
5. Click on the Raster icon in the Model Maker tool palette.
6. Click below and to the right of the Function object in the Model Maker viewer.
7. Click on the Connection icon in the Model Maker tool palette.
8. Click on the Lock icon in the Model Maker tool palette.

9. Connect the first Raster object to the Function object as an input.
10. Connect the Function object to the final Raster object as an output.
11. Click on the Selector icon in the Model Maker tool palette.
12. Click on the Lock icon in the Model Maker tool palette to turn it off.

Add Raster Properties

1. Double-click on the first Raster object.
2. Select **ortho.img** from the examples directory and click **OK**.

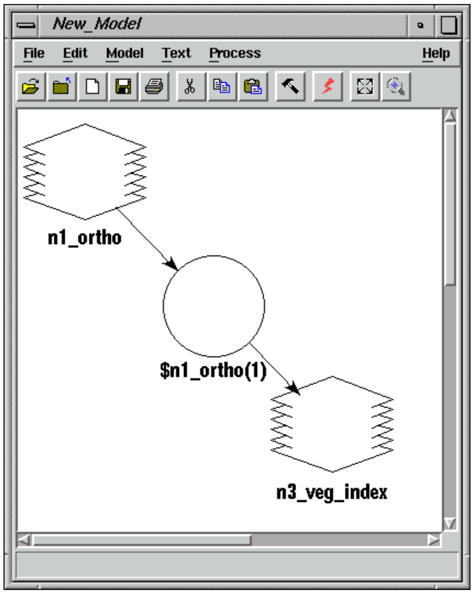
Add Function Properties

1. Double-click on the Function object.
2. Under **Available Inputs**, click on **$n1_ortho(1)**.
3. In the Calculator portion of the Function Definition dialog, click on the **/** key.
4. Under **Available Inputs**, click on **$n1_ortho(2)**.
5. Click on the **OK** button.

Add Raster Properties

1. Double-click on the output Raster object.
2. Navigate to a directory where you have write permission, and enter **veg_index.img** for the output image name.
3. Click on the **Delete If Exists** (you are going to run the model more than once to get it working properly).
4. Click on the **OK** button.

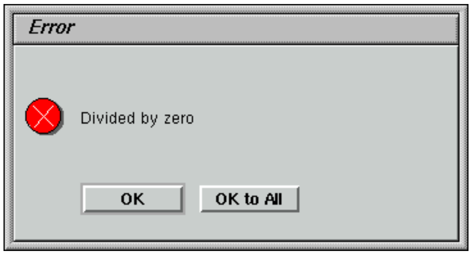
   At this point, your model should look similar to the following:


New_Model
File
Edit
Model
Text
Process
Help
n1_ortho
$n1_ortho(1)
n3_veg_index

Execute the Model

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

   After your model appears to have run to completion you receive the following error:


Error
OK
OK to All

Divided by zero

Next, you attempt to avoid dividing by zero by setting the output pixel value to zero any place there would be a division by zero.

2. Click **OK** to dismiss the Error dialog.
3. Click **OK** to dismiss the Modeler status dialog.

Change Function Properties

1. In the Model Maker viewer, double-click on the Function object.
2. Click on the **Clear** button to start our function definition from scratch.
3. In the **Functions** popup list, select **Conditional**.

4. In the list of functions, select **EITHER <arg1> IF (<test>) OR <arg2> OTHERWISE**, this should be the second item on the list.

5. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.

6. In the Calculator portion of the Function Definition dialog, click on the **0** key.

7. In the lower portion of the Function Definition dialog, click in the middle of **<test>**.

8. Under **Available Inputs**, click on **$n1_ortho(2)**.

9. In the **Functions** popup list select **Relational**.

10. In the list of functions select **==**, this should be the first item on the list.

11. In the Calculator portion of the Function Definition dialog click on the **0** key.

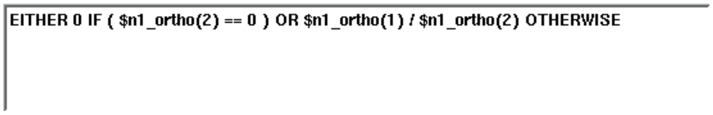
12. In the lower portion of the Function Definition dialog, click in the middle of **<arg2>**.

13. Under **Available Inputs**, click on **$n1_ortho(1)**.

14. In the Calculator portion of the Function Definition dialog, click on the **/** key.

15. Under **Available Inputs**, click on **$n1_ortho(2)**.

    Your Function Definition dialog should now contain the following:



EITHER 0 IF ($n1_ortho(2) == 0) OR $n1_ortho(1) / $n1_ortho(2) OTHERWISE

16. Click on the **OK** button.

Execute the Model

1. Select **Process | Run**, or click on the Execute the Model icon in the tool bar.

   After all the careful checking for division by zero you get the same error.The reason you still get the same error is that Modeler evaluates the entire statement at once. As it turns out, the error is generated when you do an Integer division by zero. So in order to avoid the integer division and the resulting error, you can use floating point arithmetic to set the output pixel value. You can force the use of floating point arithmetic by simply declaring our input Raster to be of type Float.

2. Click **OK** to dismiss the Error dialog.

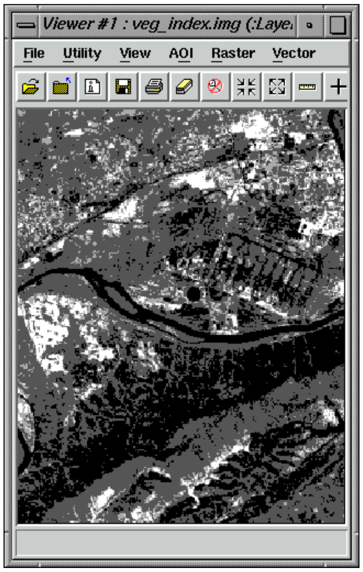
3. Click **OK** to dismiss the Modeler status dialog.

Change Raster Properties

1. In the Model Maker viewer, double-click on the input Raster object.

2. In the lower central portion of the Raster dialog, in the **Declare As** popup list select **Float**.
3. Click on the **OK** button.
4. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

   The model now runs successfully to completion without any errors. However, if you view the resulting image, **veg_index.img**, you notice that it is relatively dark and does not contain much detail.

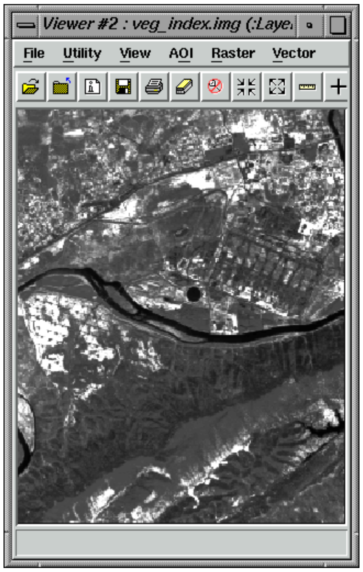


This happens because, even though you are calculating a floating-point ratio, you are still outputting an integer result. So, all resulting output pixel values are being truncated to integers. This includes all pixels where the pixel value in band two is larger than the pixel value in band one—these are all set to 0 instead of retaining values such as 0.25, 0.833, or 0.498. In order to maintain the information being calculated, all you have to do is change the type of output file being generated.

Change Raster Properties

1. In the Model Maker viewer, double-click on the output Raster object.
2. In the Raster dialog in the **Data Type** popup list select **Float Single**.
3. Click on the **OK** button.
4. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

   If you view the resulting output image now, you see the full detail from the computations, which are available for further analysis.



Use AOIs in Processing

Area Of Interest (AOI) processing can be used to restrict the area processed of individual images or of the model as a whole. AOIs can be used as masks to cookie cut the desired portions of images. When and how the mask is applied may not be of much interest in a model utilizing point operations. However, in models doing neighborhood operations, when the AOI is applied yields differing results.

For example, if you cookie cut the input image with an AOI before doing an edge detection filter, the model produces artificial edges around the AOI. In this case, you want to do the edge detection on the original input image and cookie cut the results with the AOI. Besides using AOIs as processing masks, Vector layer inputs may also be used. In the following example, you generate and use an AOI to smooth out the appearance of the water in Mobile Bay.

Create AOI

You must have Model Maker running.

1. Click on the Raster icon in the Model Maker tool palette.
2. Click near the upper left corner of the Model Maker viewer.

1. In a Viewer, open **mobbay.img** from the /examples directory.
2. In the Viewer, click on the Show Tool Palette for Top Layer icon.
3. Select the Region Grow AOI tool from the palette.
4. Click a dark portion of the water near the southeast corner of the image.
5. From the **AOI** menu select **Seed Properties**.
6. Click the **Area** checkbox to turn it off.
7. Enter **20.0** for the **Spectral Euclidean Distance**.

8. Click on the **Redo** button.

Add Raster Properties

1. Double-click on the first Raster object.
2. Select **mobbay.img** from the /examples directory.
3. Click on the **Choose AOI** button on the right side of the Raster dialog.
4. Select **Viewer** as the **AOI Source** and click **OK**.
5. Click **OK** in the Raster dialog.

Add Raster Properties

1. Click on the Raster icon in the Model Maker tool palette.
2. Click to the right of the existing Raster in the Model Maker viewer.
3. Double-click this newly placed Raster object.
4. Select **mobbay.img** from the /examples directory.
5. This time do not select an AOI, but rather just click **OK** in the Raster dialog.

Add Matrix Properties

1. Click on the Matrix icon in the Model Maker tool palette.
2. Click just to the right of the two Raster objects in the Model Maker viewer.
3. Double-click this newly placed Matrix object.
4. In the **Size** popup list select **5** × **5**.
5. Click **OK**.

Add Function Properties

1. Click on the Function icon in the Model Maker tool palette.
2. Click below **n3_Low_Pass** in the Model Maker viewer.
3. Connect **n2_mobbay** and **n3_Low_Pass** to the newly placed Function object.
4. Double-click on the Function object.
5. From the analysis **Functions** select **CONVOLVE (<raster> , <kernel>)**, this should be the third item on the list.

6. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.
7. Under **Available Inputs**, click on **$n2_mobbay**.
8. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.
9. Under **Available Inputs**, click on **$n3_Low_Pass**.
10. Click **OK**.

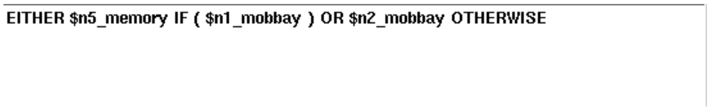
Add Raster Properties

1. Click on the Raster icon in the Model Maker tool palette.
2. Click below the Function object in the Model Maker viewer.
3. Connect the Function object to the new Raster object.
4. Double-click the new Raster object.
5. Click **Temporary Raster Only**.
6. Click **OK**.

Add Function Properties

1. Click on the Function icon in the Model Maker tool palette.
2. Click to the left of **n5_memory** in the Model Maker viewer.
3. Connect **n1_mobbay**, **n2_mobbay**, and **n5_memory** to the Function object.
4. Double-click on the Function object.
5. In the **Functions** popup list select **Conditional**.
6. In the list of functions select **EITHER <arg1> IF (<test>) OR <arg2> OTHERWISE**, this should be the second item on the list.
7. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.
8. Under **Available Inputs**, click on **$n5_memory**.
9. In the lower portion of the Function Definition dialog, click in the middle of **<test>**.
10. Under **Available Inputs**, click on **$n1_mobbay**.
11. In the lower portion of the Function Definition dialog, click in the middle of **<arg2>**.
12. Under **Available Inputs**, click on **$n2_mobbay**.

    Your Function Definition dialog should now contain the following:



EITHER $n5_memory IF ($n1_mobbay) OR $n2_mobbay OTHERWISE

$n5_memory is the filtered image, **$n1_mobbay** has the AOI mask, and **$n2_mobbay** is the original image.

13. Click **OK**.

Add Raster Properties

1. Click on the Raster icon in the Model Maker tool palette.

2. Click below the new Function object in the Model Maker viewer.

3. Connect the Function object to the Raster object.

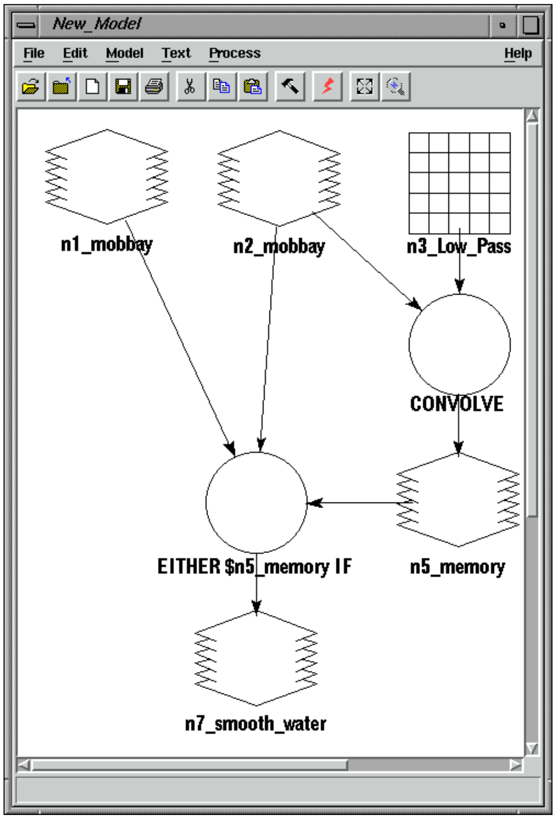
4. Double-click on the output Raster object.

5. Navigate to a directory where you have write permission and enter **smooth_water.img** for the output image name.

6. Click on the **Delete If Exists**, in case you need to run the model more than once to get it working properly.

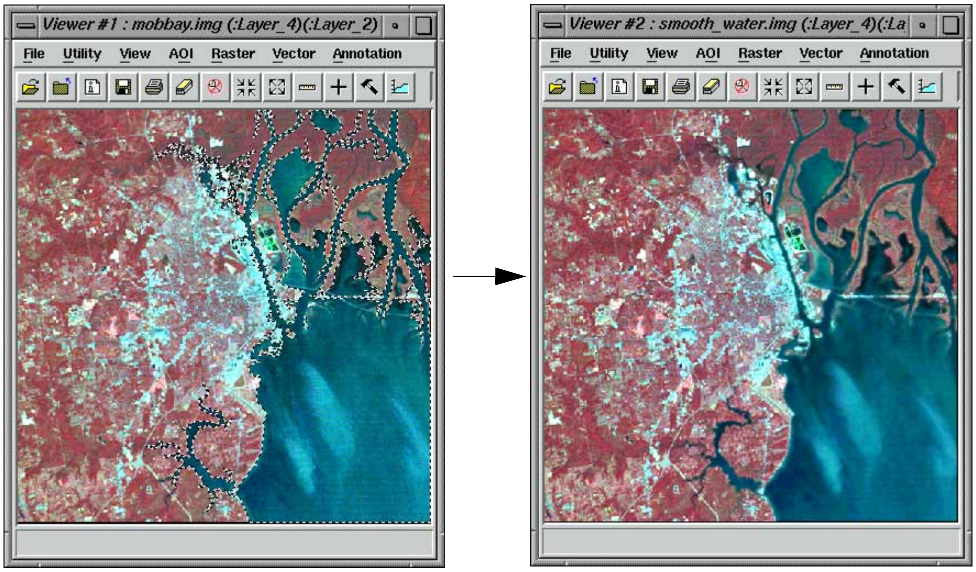
7. Click on the **OK** button.

Your model should look like the following:


New_Model
File
Edit
Model
Text
Process
Help
n1_mobbay
n2_mobbay
n3_Low_Pass
CONVOLVE
EITHER $n5_memory IF
n5_memory
n7_smooth_water

Execute the Model

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.
2. Use the Viewer and the Swipe tool to compare the original **mobbay.img** and the new **smooth_water.img**.


Viewer #1 : mobbay.img (:Layer_4)(:Layer_2)
File Utility View AOI Raster Vector Annotation
Viewer #2 : smooth_water.img (:Layer_4)(:La
File Utility View AOI Raster Vector Annotation

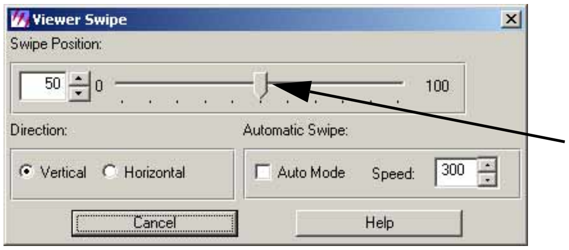
Compare the image on the left, above, with the image on the right. The image on the right has been visibly smoothed in the areas of water. You can see this more clearly if you use the Swipe utility in the Viewer (see below).

Using the Swipe Utility

1. Open one file in the Viewer

2. Open the second file in the Viewer making sure to uncheck the **Clear Display** checkbox in the **Raster Options** tab of the file selector.

3. Select **Swipe** from the Viewer **Utility** menu. The Viewer Swipe dialog opens.


Viewer Swipe

Swipe Position: 50

Direction: ● Vertical ○ Horizontal

Automatic Swipe: ☐ Auto Mode Speed: 300

0
100
Cancel
Help

Drag this slider

4. Grab the **Swipe Position** slider and drag it left and right while observing the Viewer. As you move the slider to the left, the top image is rolled back to reveal the underlying image.

CHAPTER SEVENTEEN

Advanced Classification

Introduction

Classification is the process of sorting pixels into a finite number of individual classes, or categories of data based on their data file values. If a pixel satisfies a certain set of criteria, then the pixel is assigned to the class that corresponds to that criteria.

There are two ways to classify pixels into different categories:

- supervised
- unsupervised

Supervised vs. Unsupervised Classification

Supervised classification is more closely controlled by you than unsupervised classification. In this process, you select pixels that represent patterns you recognize or that you can identify with help from other sources. Knowledge of the data, the classes desired, and the algorithm to be used is required before you begin selecting training samples.

By identifying patterns in the imagery, you can train the computer system to identify pixels with similar characteristics. By setting priorities to these classes, you supervise the classification of pixels as they are assigned to a class value. If the classification is accurate, then each resulting class corresponds to a pattern that you originally identified.

Unsupervised classification is more computer-automated. It allows you to specify parameters that the computer uses as guidelines to uncover statistical patterns in the data.

In this tour guide, you perform both a supervised and an unsupervised classification of the same image file.

All of the data used in this tour guide are in the <IMAGINE_HOME>/examples directory. You should copy the **germtm.img** *file to a different directory so that you can have write permission to this file.*

Approximate completion time for this tour guide is 2 hours.

Perform Supervised Classification

This section shows how the Supervised Classification tools allow you to control the classification process.

You perform the following operations in this section:

- define signatures
- evaluate signatures
- process a supervised classification

Define Signatures using Signature Editor

The ERDAS IMAGINE Signature Editor allows you to create, manage, evaluate and edit signatures (.sig extension). The following types of signatures can be defined:

- parametric (statistical)
- nonparametric (feature space)

In this section, you define the signatures using the following operations:

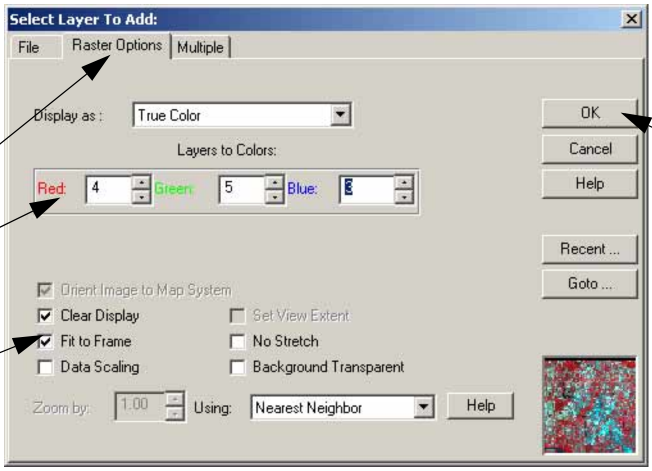
- collect signatures from the image to be classified using the area of interest (AOI) tools
- collect signatures from the Feature Space image using the AOI tools and Feature Space tools

Preparation

ERDAS IMAGINE must be running and a Viewer must be open.

1. Select **File | Open | Raster Layer** from the Viewer menu bar or click the Open icon on the Viewer tool bar to display the image file to be classified.

   The Select Layer To Add dialog opens.



Display as : True Color

Layers to Colors:

Red: 4 Green: 5 Blue: 3

☑ Orient Image to Map System

☑ Clear Display ☐ Set View Extent

☑ Fit to Frame ☐ No Stretch

☐ Data Scaling ☐ Background Transparent

Zoom by: 1.00 Using: Nearest Neighbor

Click here to select the raster options

Set values to 4, 5, 3

Click on Fit to Frame

Click here to display the image

2. In the Select Layer To Add dialog under **Filename**, select **germtm.img**. This is the image file that is going to be classified.

3. Click on the **Raster Options** tab at the top of the dialog, and then set the **Layers to Colors** to **4**, **5**, and **3** (red, green, and blue, respectively).

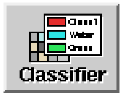
4. Click the **Fit to Frame** option to enable it.

5. Click **OK** in the Select Layer To Add dialog.

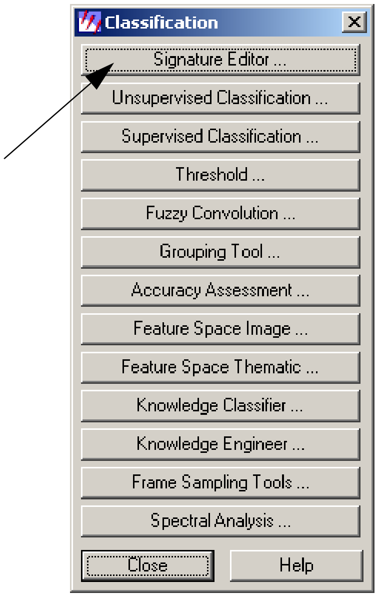
   The file **germtm.img** displays in the Viewer.

Open Signature Editor

1. Click on the Classifier icon on the ERDAS IMAGINE icon panel.


Class 1
Water
Grass
Classifier

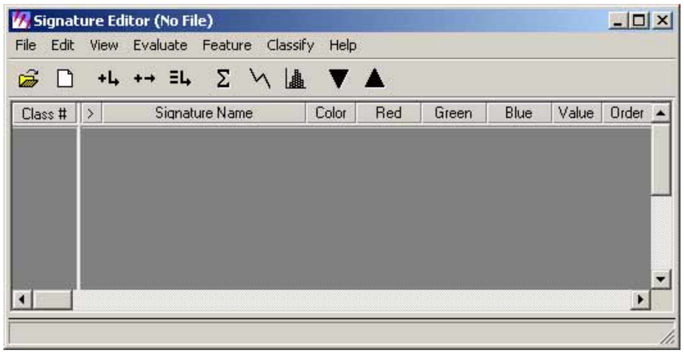
   The **Classification** menu displays.


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Grouping Tool ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Frame Sampling Tools ...
Spectral Analysis ...
Close
Help

Click here to start the Signature Editor

2. Select **Signature Editor** from the **Classification** menu to start the Signature Editor.

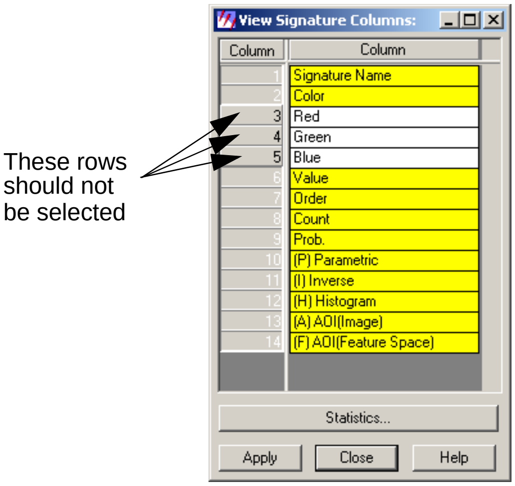
   The Signature Editor opens.


Signature Editor (No File)
File Edit View Evaluate Feature Classify Help
Class # > Signature Name Color Red Green Blue Value Order

3. In the **Classification** menu, click **Close** to remove this menu from the screen.

4. In the Signature Editor, select **View | Columns**.

   The View Signature Columns dialog opens.


View Signature Columns:
Column
Column
1 Signature Name
2 Color
3 Red
4 Green
5 Blue
6 Value
7 Order
8 Count
9 Prob.
10 (P) Parametric
11 (I) Inverse
12 (H) Histogram
13 (A) AOI(Image)
14 (F) AOI(Feature Space)
Statistics...
Apply
Close
Help

These rows should not be selected

5. In the View Signature Columns dialog, right-click in the first column to access the Row Selection menu. Click on Select All.

6. Shift-click **Red**, **Green**, and **Blue** in **Column** boxes **3**, **4**, and **5** to deselect these rows.

   These are the CellArray columns in the Signature Editor that you remove to make it easier to use. These columns can be returned at any time.

7. In the View Signature Columns dialog, click **Apply**.

   The **Red**, **Green**, and **Blue** columns are deleted from the Signature Editor.

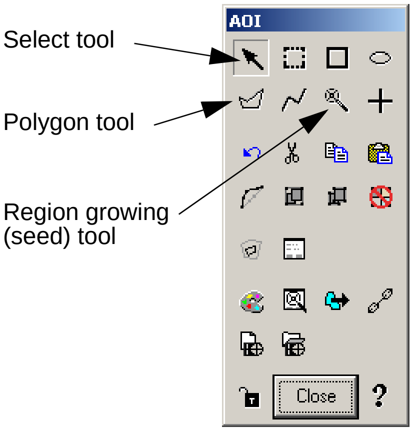
8. Click **Close** in the View Signature Columns dialog.

Use AOI Tools to Collect Signatures

The AOI tools allow you to select the areas in an image to be used as a signature. These signatures are parametric because they have statistical information.

1. Select **AOI | Tools** from the Viewer menu bar.

   The AOI tool palette displays.

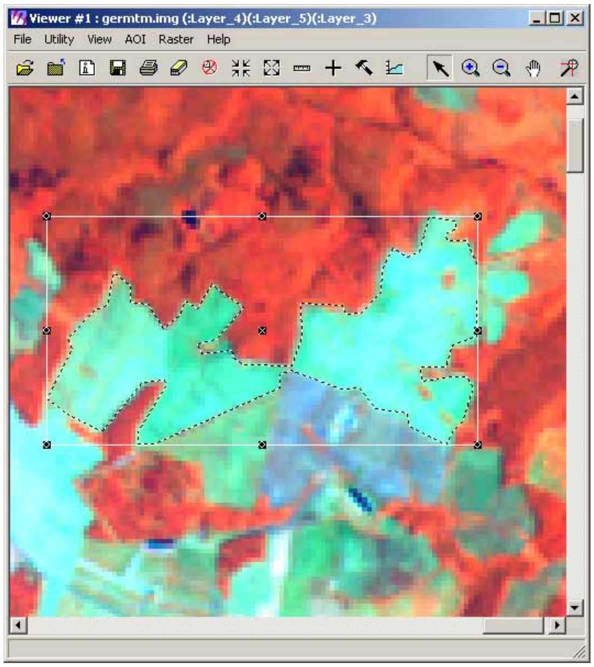


2. Use the Zoom In tool on the Viewer tool bar to zoom in on one of the light green areas in the **germtm.img** file in the Viewer.

3. In the AOI tool palette, click on the Polygon icon .

4. In the Viewer, draw a polygon around the green area you just magnified. Click and drag to draw the polygon and click to draw the vertices. Middle-click or double-click to close the polygon (depending on what is set in **Session | Preferences**).

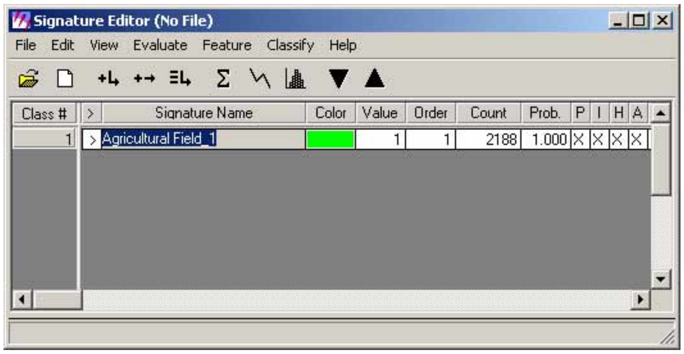
   After the AOI is created, a bounding box surrounds the polygon, indicating that it is currently selected. These areas are agricultural fields.


Viewer #1 : germtm.img (:Layer_4)(:Layer_5)(:Layer_3)
File Utility View AOI Raster Help

5. In the Signature Editor, click on the Create New Signature(s) from AOI icon or select **Edit | Add** from the menu bar to add this AOI as a signature.

6. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Agricultural Field_1,** then press Return on the keyboard.

7. In the Signature Editor, hold in the **Color** column next to **Agricultural Field_1** and select **Green**.


Signature Editor (No File)
File Edit View Evaluate Feature Classify Help

Class #	>	Signature Name	Color	Value	Order	Count	Prob.	P	I	H	A
1	>	Agricultural Field_1	🟩	1	1	2188	1.000	☒	☒	☒	☒

8. Zoom in on one of the light blue/cyan areas in the **germtm.img** file in the Viewer.

9. Draw a polygon as you did in steps 2. through step 4.

   These areas are also agricultural fields.

10. After the AOI is created, a bounding box surrounds the polygon, indicating that it is currently selected. In the Signature Editor, click on the Create New Signature(s) from AOI icon or select **Edit | Add** to add this AOI as a signature.

11. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Agricultural Field_2**, then press Return on the keyboard.

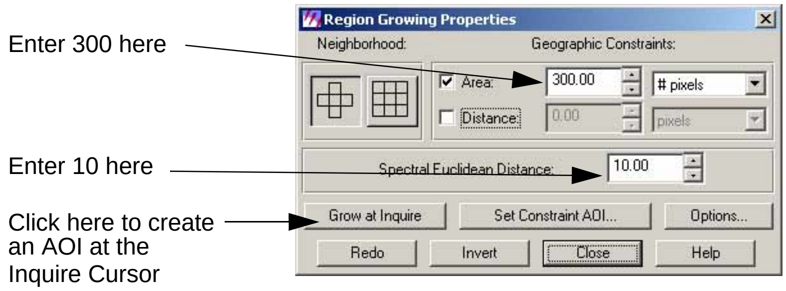
12. In the Signature Editor, hold in the **Color** column next to **Agricultural Field_2** and select **Cyan**.

Select Neighborhood Options

This option determines which pixels are considered contiguous (i.e., similar values) to the seed pixel or any accepted pixels.

1. Select **AOI | Seed Properties** from the Viewer menu bar.

   The Region Growing Properties dialog opens.



Region Growing Properties

Neighborhood:

Geographic Constraints:

☑ Area: 300.00 # pixels

☐ Distance: 0.00 pixels

Spectral Euclidean Distance: 10.00

Enter 300 here

Enter 10 here

Click here to create an AOI at the Inquire Cursor

2. Click on the Neighborhood icon in the Region Growing Properties dialog.

   This option specifies that four pixels are to be searched. Only those pixels above, below, to the left, and to the right of the seed or any accepted pixels are considered contiguous.

3. Under **Geographic Constraints**, the **Area** checkbox should be turned on to constrain the region area in pixels. Enter **300** into the **Area** number field and press Return on your keyboard.

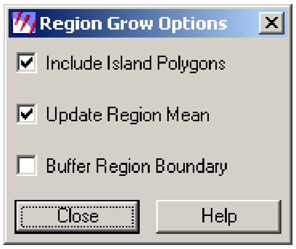
   This is the maximum number of pixels that are in the AOI.

4. Enter **10.00** in the **Spectral Euclidean Distance** number field.

   The pixels that are accepted in the AOI are within this spectral distance from the mean of the seed pixel.

5. Next, click on **Options** in the Region Growing Properties dialog.

   The Region Grow Options dialog opens.


Region Grow Options
Close
Help

☑ Include Island Polygons

☑ Update Region Mean

☐ Buffer Region Boundary

6. In the Region Grow Options dialog, make sure that the **Include Island Polygons** checkbox is turned on in order to include polygons in the growth region.

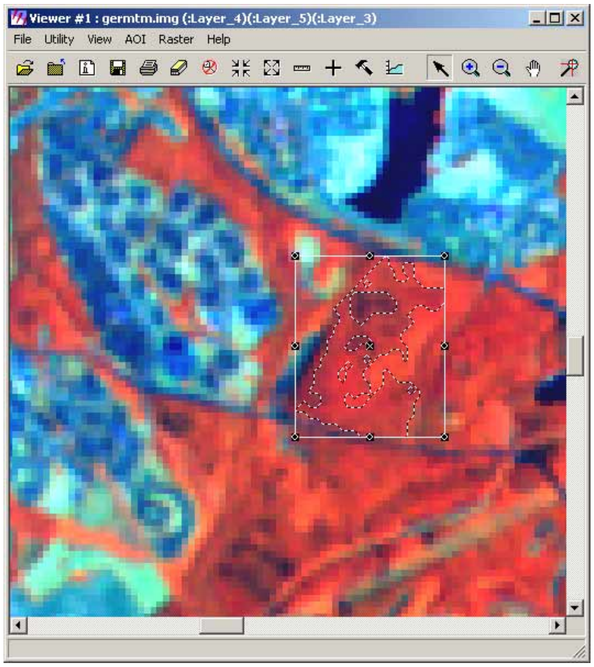
7. Click **Close** in the Region Grow Options dialog.

Create an AOI

1. In the AOI tool palette, click on the Region Grow icon .

2. Click inside a bright red area in the **germtm.img** file in the Viewer.

   This is a forest area. A polygon opens and a bounding box surrounds the polygon, indicating that it is selected.


Viewer #1 : germtm.img (:Layer_4)(:Layer_5)(:Layer_3)
File
Utility
View
AOI
Raster
Help

3. In the Region Growing Properties dialog, enter new numbers in the **Area** and **Spectral Euclidean Distance** number fields (e.g., **500** for **Area** and **15** for **Spectral Euclidean Distance**) to see how this modifies the AOI polygon.

4. In the Region Growing Properties dialog, click **Redo** to modify the AOI polygon with the new parameters.

Add a Signature

1. After the AOI is created, click on the Create New Signature(s) from AOI icon +↳ in the Signature Editor to add this AOI as a signature.
2. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Forest_1**, then press Return on the keyboard.
3. In the Signature Editor, hold in the **Color** column next to **Forest_1** and select **Yellow**.
4. In the Region Growing Properties dialog, enter **300** in the **Area** number field.

Add Another Signature

1. In the Viewer, select **Utility | Inquire Cursor**.

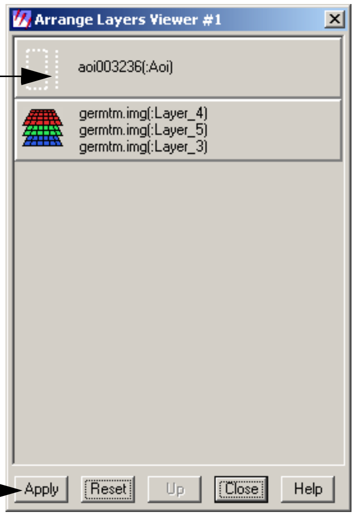
   The Inquire Cursor dialog opens and the inquire cursor (a white crosshair) is placed in the Viewer. The inquire cursor allows you to move to a specific pixel in the image and use it as the seed pixel.

2. Drag the intersection of the inquire cursor to a dark red area in the **germtm.img** file in the Viewer. This is also a forest area.
3. In the Region Growing Properties dialog, click **Grow at Inquire**. Wait for the polygon to open.
4. After the AOI is created, click on the Create New Signature(s) from AOI icon +↳ in the Signature Editor to add this AOI as a signature.
5. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Forest_2**, then press Return on the keyboard.
6. In the Signature Editor, hold in the **Color** column next to **Forest_2** and select **Pink**.
7. Click **Close** in the Inquire Cursor dialog and the Region Growing Properties dialog.

Arrange Layers

1. Now that you have the parametric signatures collected, you do not need the AOIs in the Viewer. Select **View | Arrange Layers** from the Viewer menu bar.

   The Arrange Layers dialog opens.


Arrange Layers Viewer #1
aoi003236(:Aoi)
germtm.img(:Layer_4)
germtm.img(:Layer_5)
germtm.img(:Layer_3)
Apply
Reset
Up
Close
Help

Right-hold over this button to delete the AOI layer

Click here to apply the changes you have made to the Viewer

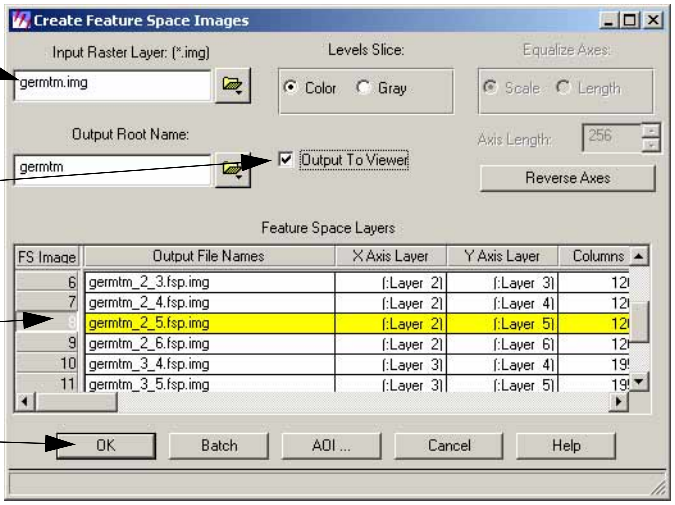
2. In the Arrange Layers dialog, right-hold over the **AOI Layer** button and select **Delete Layer** from the **AOI Options** menu.
3. Click **Apply** in the Arrange Layers dialog to delete the AOI layer.
4. You are asked if you want to save the changes before closing. Click **No**.
5. In the Arrange Layers dialog, click **Close**.

Create Feature Space Image

The ERDAS IMAGINE Feature Space tools allow you to interactively define areas of interest (polygons or rectangles) in the Feature Space image(s). A Feature Space signature (nonparametric) is based on the AOI(s) in the Feature Space image. Use this technique to extract a signature for water.

1. Select **Feature | Create | Feature Space Layers** from the Signature Editor menu bar.

   The Create Feature Space Images dialog opens.



Input Raster Layer: germtm.img

Levels Slice: ● Color ○ Gray

Equalize Axes: ● Scale ○ Length

Output Root Name: germtm

☑ Output To Viewer

Axis Length: 256

FS Image	Output File Names	X Axis Layer	Y Axis Layer	Columns
6	germtm_2_3.fsp.img	(:Layer_2)	(:Layer_3)	12
7	germtm_2_4.fsp.img	(:Layer_2)	(:Layer_4)	12
8	germtm_2_5.fsp.img	(:Layer_2)	(:Layer_5)	12
9	germtm_2_6.fsp.img	(:Layer_2)	(:Layer_6)	12
10	germtm_3_4.fsp.img	(:Layer_3)	(:Layer_4)	19
11	germtm_3_5.fsp.img	(:Layer_3)	(:Layer_5)	19

Enter germtm.img

Click here to
output to a Viewer

Click on
germtm_2_5.fsp.img

Click here to
create image

2. In the Create Feature Space Images dialog under **Input Raster Layer**, enter **germtm.img**. This is the image file from which the Feature Space image is generated.

   Under **Output Root Name**, the default name is **germtm**. This is the root name for the Feature Space image files that are generated.

   Verify that the directory where the Feature Space image files are saved has write permission.

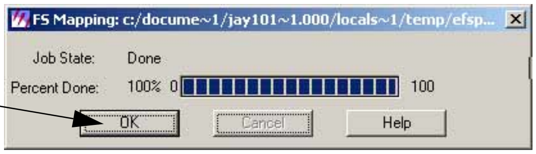
3. In the Create Feature Space Images dialog, click the **Output to Viewer** option so that the Feature Space image is displayed in a Viewer.

4. Under **Feature Space Layers**, click the number **8** in the **FS Image** column in the CellArray to select the **germtm_2_5.fsp.img** row. (You may need to scroll down to get to **FS Image** number **8**.)

   The output Feature Space image is based on layers two and five of the **germtm.img** file. Layers two and five are selected since water is spectrally distinct in this band combination.

5. Click **OK** in the Create Feature Space Images dialog to create the Feature Space image for layers two and five of the **germtm.img** file.

   The Create Feature Space Images dialog closes, then the Job Status dialog opens.



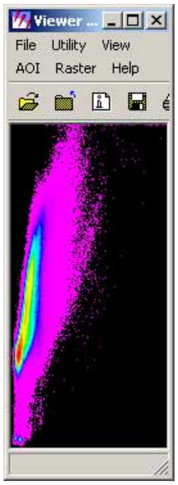
FS Mapping: c:/docume~1/jay101~1.000/locals~1/temp/efsp...

Job State: Done

Percent Done: 100% 0 100

Click here to close
this dialog

After the process is completed, a Viewer (Viewer #2) opens, displaying the Feature Space image.


Viewer ...
File Utility View
AOI Raster Help

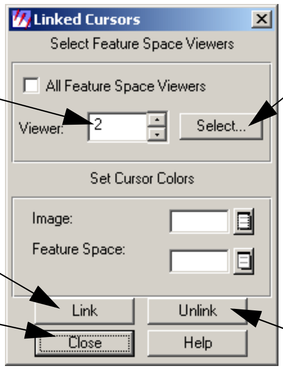
6. Click **OK** in the Job Status dialog to close this dialog.

Link Cursors in Image/Feature Space

The Linked Cursors utility allows you to directly link a cursor in the image Viewer to the Feature Space viewer. This shows you where pixels in the image file are located in the Feature Space image.

1. In the Signature Editor, select **Feature | View | Linked Cursors**.

The Linked Cursors dialog opens.


Linked Cursors
Select Feature Space Viewers
☐ All Feature Space Viewers
Viewer: 2
Select...
Set Cursor Colors
Image:
Feature Space:
Link
Unlink
Close
Help

This is set to 2

Click to select the Feature Space viewer

Click to link the Viewers

Click to close this dialog

Click to unlink Viewers

2. Click **Select** in the Linked Cursors dialog to define the Feature Space viewer that you want to link to the Image Viewer.

3. Click in Viewer #2 (the Viewer displaying the Feature Space image).

The **Viewer** number field in the Linked Cursors dialog changes to **2**. You could also enter a **2** in this number field without having to click the **Select** button.

4. In the Linked Cursors dialog, click **Link** to link the Viewers, then click in the Viewer displaying **germtm.img**.

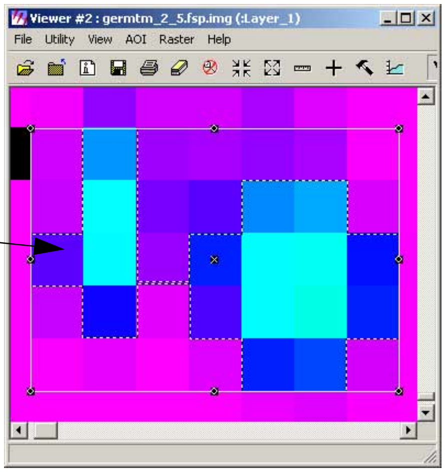
   The linked inquire cursors (white crosshairs) open in the Viewers.

5. Drag the inquire cursor around in the **germtm.img** Viewer (Viewer #1) to see where these pixels are located in the Feature Space image. Notice where the water areas are located in the Feature Space image. These areas are black in the **germtm.img** file (Viewer #1).

Define Feature Space Signature

Any Feature Space AOI can be defined as a nonparametric signature in your classification.

1. Use the polygon AOI tool to draw a polygon in the Feature Space image. Draw the polygon in the area that you identified as water. You may need to zoom in on the images (i.e., zoom in on either or both the Feature Space image or **germtm.img**) to draw the polygon. The Feature Space signature is based on this polygon.


Viewer #2 : germtm_2_5.fsp.img (:Layer_1)
File Utility View AOI Raster Help

Draw a polygon in the area identified as water

2. After the AOI is created, click on the Create New Signature(s) from AOI icon in the Signature Editor to add this AOI as a signature.

3. The signature you have just added is a nonparametric signature. Select **Feature | Statistics** from the Signature Editor menu bar to generate statistics for the Feature Space AOI.

   A Job Status dialog displays, stating the progress of the function.

4. When the function is 100% complete, click **OK** in the Job Status dialog.

   The Feature Space AOI now has parametric properties.

5. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Water**, then press the Return key on the keyboard.

6. In the Signature Editor, hold in the **Color** column next to **Water** and select **Blue**.

7. In the Linked Cursors dialog, click **Unlink** to unlink the viewers.

   The inquire cursors are removed from the viewers.

8. In the Linked Cursors dialog, click **Close**.

9. Now that you have the nonparametric signature collected, you do not need the AOI in the Feature Space viewer. Select **View | Arrange Layers** from the Viewer #2 menu bar.

   The Arrange Layers dialog opens.

10. In the Arrange Layers dialog, right-hold over the **AOI Layer** button and select **Delete Layer** from the AOI Options popup list.

11. Click **Apply** in the Arrange Layers dialog to delete the AOI layer.

12. You are asked if you want to save the changes before closing. Click **No**.

13. In the Arrange Layers dialog, click **Close**.

14. Practice taking additional signatures using any of the signature-generating techniques you have learned in the steps above. Extract at least five signatures.

15. After you have extracted all the signatures you wish, select **File | Save As** from the Signature Editor menu bar.

    The Save Signature File As dialog opens.

16. Use the Save Signature File As dialog to save the signature set in the Signature Editor (e.g., **germtm_siged.sig**).

17. Click **OK** in the Save Signature File As dialog.

Use Tools to Evaluate Signatures

Once signatures are created, they can be evaluated, deleted, renamed, and merged with signatures from other files. Merging signatures allows you to perform complex classifications with signatures that are derived from more than one training method (supervised and/or unsupervised, parametric and nonparametric).

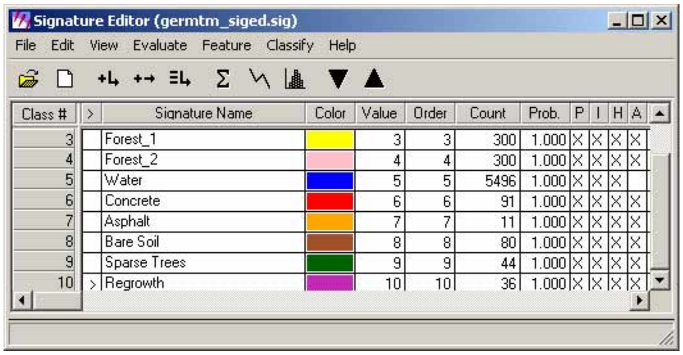
Next, the following tools for evaluating signatures are discussed:

- alarms
- contingency matrix
- feature space to image masking
- signature objects
- histograms
- signature separability
- statistics

When you use one of these tools, you need to select the appropriate signature(s) to be used in the evaluation. For example, you cannot use the signature separability tool with a nonparametric (Feature Space) signature.

Preparation

You should have at least ten signatures in the Signature Editor, similar to the following:


Signature Editor (germtm_siged.sig)
File Edit View Evaluate Feature Classify Help

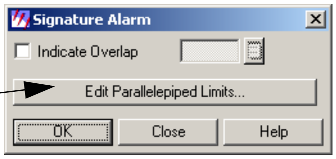
Class #	>	Signature Name	Color	Value	Order	Count	Prob.	P	I	H	A
3		Forest_1	🟨	3	3	300	1.000	X	X	X	X
4		Forest_2	▨	4	4	300	1.000	X	X	X	X
5		Water	🟦	5	5	5496	1.000	X	X	X	
6		Concrete	🟥	6	6	91	1.000	X	X	X	X
7		Asphalt	🟧	7	7	11	1.000	X	X	X	X
8		Bare Soil	🟫	8	8	80	1.000	X	X	X	X
9		Sparse Trees	🟩	9	9	44	1.000	X	X	X	X
10	>	Regrowth	🟪	10	10	36	1.000	X	X	X	X

Set Alarms

The Signature Alarm utility highlights the pixels in the Viewer that belong to, or are estimated to belong to a class according to the parallelepiped decision rule. An alarm can be performed with one or more signatures. If you do not have any signatures selected, then the active signature, which is next to the **>**, is used.

1. In the Signature Editor, select **Forest_1** by clicking in the **>** column for that signature. The alarm is performed with this signature.
2. In the Signature Editor menu bar, select **View | Image Alarm**.

   The Signature Alarm dialog opens.



☐ Indicate Overlap

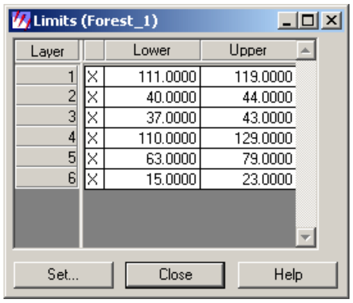
Click to change the parallelepiped limits

3. Click **Edit Parallelepiped Limits** in the Signature Alarm dialog to view the limits for the parallelepiped.

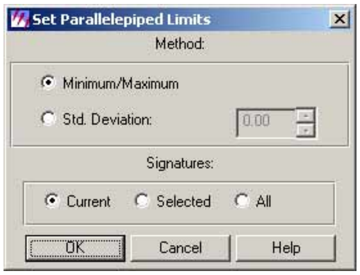
   The Limits dialog opens.

4. In the Limits dialog, click **Set** to define the parallelepiped limits.

   The Set Parallelepiped Limits dialog opens.



Layer		Lower	Upper
1	☒	111.0000	119.0000
2	☒	40.0000	44.0000
3	☒	37.0000	43.0000
4	☒	110.0000	129.0000
5	☒	63.0000	79.0000
6	☒	15.0000	23.0000



Method:

● Minimum/Maximum

○ Std. Deviation: 0.00

Signatures:

● Current ○ Selected ○ All

The Signature Alarm utility allows you to define the parallelepiped limits by either:

- the minimum and maximum for each layer in the signature, or
- a specified number of standard deviations from the mean of the signature.

5. If you wish, you can set new parallelepiped limits and click **OK** in the Set Parallelepiped Limits dialog, or simply accept the default limits by clicking **OK** in the Set Parallelepiped Limits dialog.

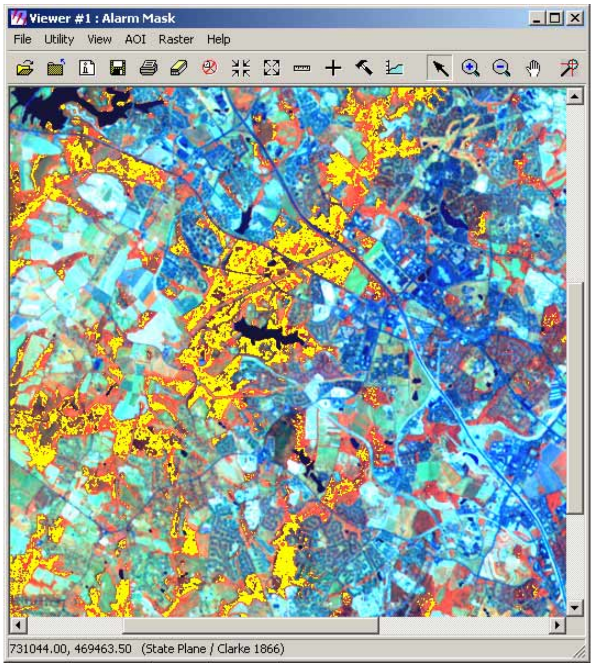
   The new/default limits display in the Limits CellArray.

6. Click **Close** in the Limits dialog.

7. In the Signature Alarm dialog, click **OK**.

   The alarmed pixels display in the Viewer in yellow. You can use the toggle function (**Utility | Flicker**) in the Viewer to see how the pixels are classified by the alarm.

Be sure that there are no AOI layers open on top of the Alarm Mask Layer. You can use **View | Arrange Layers** *to remove any AOI layers present in the Viewer.*


Viewer #1 : Alarm Mask
File Utility View AOI Raster Help
731044.00, 469463.50 (State Plane / Clarke 1866)

8. In the Signature Alarm dialog, click **Close**.

9. In the Viewer #1 menu bar, select **View | Arrange Layers**.

The Arrange Layers dialog opens.

10. In the Arrange Layers dialog, right-hold over the **Alarm Mask** button and select **Delete Layer** from the **Layer Options** menu.

11. Click **Apply** to delete the alarm layer from the Viewer.

12. You are asked if you want to save the changes before closing. Click **No**.

13. In the Arrange Layers dialog, click **Close**.

Evaluate Contingency

Contingency Matrix

The Contingency Matrix utility allows you to evaluate signatures that have been created from AOIs in the image. This utility classifies only the pixels in the image AOI training sample, based on the signatures. It is usually expected that the pixels of an AOI are classified to the class that they train. However, the pixels of the AOI training sample only weight the statistics of the signature. They are rarely so homogenous that every pixel actually becomes assigned to the expected class. The Contingency Matrix utility can be performed with multiple signatures. If you do not have any signatures selected, then all of the signatures are used.

The output of the Contingency Matrix utility is a matrix of percentages that allows you to see how many pixels in each AOI training sample were assigned to each class. In theory, each AOI training sample would be composed primarily of pixels that belong to its corresponding signature class.

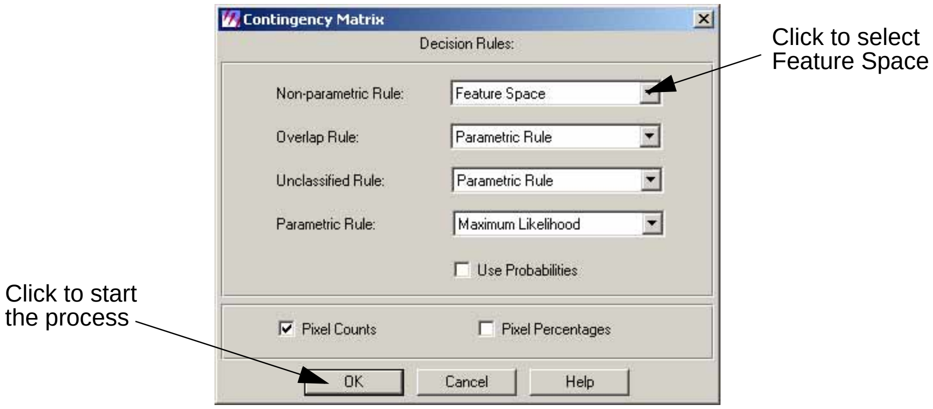
The AOI training samples are classified using one of the following classification algorithms:

- parallelepiped
- feature space
- maximum likelihood
- Mahalanobis distance

NOTE: The classification decision rule that is going to be used in the actual image classification should be determined so that it can also be used in the Contingency Matrix utility.

1. In the Signature Editor, select all of the signatures by Shift-clicking in the first row of the **Class** column and then dragging down through the other classes.
2. In the Signature Editor menu bar, select **Evaluate | Contingency**.

   The Contingency Matrix dialog opens.



Contingency Matrix

Decision Rules:

Non-parametric Rule: Feature Space

Overlap Rule: Parametric Rule

Unclassified Rule: Parametric Rule

Parametric Rule: Maximum Likelihood

☐ Use Probabilities

☑ Pixel Counts ☐ Pixel Percentages

Click to select Feature Space

Click to start the process

3. In the Contingency Matrix dialog, click on the **Non-parametric Rule** popup list and select **Feature Space**.

See "Chapter 6: Classification" *of the* ERDAS Field Guide *for more information on decision rules.*

4. Click **OK** in the Contingency Matrix dialog to start the process.

   A Job Status dialog displays, stating the progress of the function.

5. When the process is 100% complete, click **OK** in the Job Status dialog.

   The IMAGINE Text Editor opens (labelled **Editor, Dir**), displaying the error matrix.

6. After viewing the reference data in the Text Editor, select **File | Close** from the menu bar.

7. Deselect the signatures that were selected by right-clicking in the **Class** column and choosing **Select None** from the **Row Selection** menu.

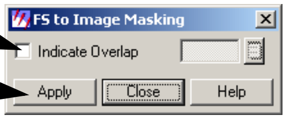
Generate a Mask from a Feature Space Signature

The Feature Space to Image Masking utility allows you to generate a mask from a Feature Space signature (i.e., the AOI in the Feature Space image). Once the Feature Space signature is defined as a mask, the pixels under the mask are identified in the image file and highlighted in the Viewer. This allows you to view which pixels would be assigned to the Feature Space signature's class. A mask can be generated from one or more Feature Space signatures. If you do not have any signatures selected, then the active signature, which is next to the **>**, is used.

The image displayed in Viewer #1 must be the image from which the Feature Space image was created.

1. In the Signature Editor, select **Feature | Masking | Feature Space to Image**.

   The FS to Image Masking dialog opens.


FS to Image Masking
☐ Indicate Overlap
Apply
Close
Help

This checkbox should be turned off (disabled)

Click to create the mask

2. In the Signature Editor, click in the **Class** row for **Water** to select that signature.

   The mask is generated from this Feature Space signature.

3. Disable the Indicate Overlap checkbox, and click **Apply** in the FS to Image Masking dialog to generate the mask in Viewer #1.

   A mask is placed in the Viewer.

4. In the FS to Image Masking dialog, click **Close**.

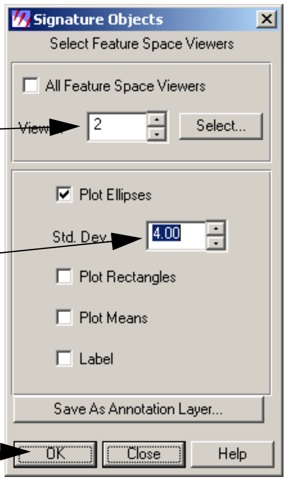
5. Deselect the **Water** feature.

View Signature Objects

The Signature Objects dialog allows you to view graphs of signature statistics so that you can compare signatures. The graphs display as sets of ellipses in a Feature Space image. Each ellipse is based on the mean and standard deviation of one signature. A graph can be generated for one or more signatures. If you do not have any signatures selected, then the active signature, which is next to the **>**, is used. This utility also allows you to show the mean for the signature for the two bands, a parallelepiped, and a label.

1. In the Signature Editor menu bar, select **Feature | Objects**.

   The Signature Objects dialog opens.


Signature Objects
Select Feature Space Viewers
☐ All Feature Space Viewers
Select...
Save As Annotation Layer...
OK
Close
Help

View: 2

☑ Plot Ellipses

Std. Dev: 4.00

☐ Plot Rectangles

☐ Plot Means

☐ Label

Enter 2 here

Enter 4 here

Click here to start the process

2. In the Signature Editor, select the signatures for **Agricultural Field_1** and **Forest_1** by clicking in the **Class** row for **Agricultural Field_1** and Shift-clicking in the **Class** row for **Forest_1**.

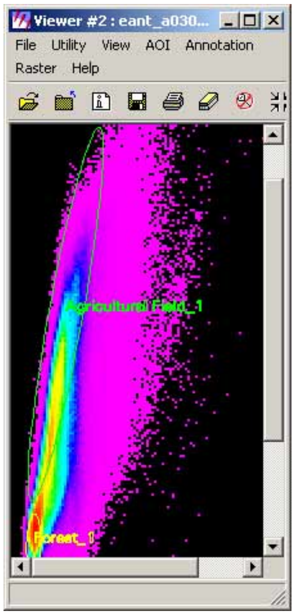
3. In the Signature Objects dialog, confirm that the **Viewer** number field is set for **2**.

4. Set the **Std. Dev.** number field to **4**.

5. Enable the **Label** checkbox by clicking on it.

6. Click **OK** in the Signature Objects dialog.

The ellipses for the **Agricultural Field_1** and **Forest_1** signatures are displayed in the Feature Space viewer.



Compare Ellipses

By comparing the ellipses for different signatures for a one band pair, you can easily see if the signatures represent similar groups of pixels by seeing where the ellipses overlap on the Feature Space image.

- When ellipses do not overlap, the signatures represent a distinct set of pixels in the two bands being plotted, which is desirable for classification. However, some overlap is expected, because it is rare that all classes are totally distinct.

When the ellipses do overlap, then the signatures represent similar pixels, which is not desirable for classification.

7. In the Signature Objects dialog, click **Close.**

8. Deselect the signatures for **Agricultural Field_1** and **Forest_1**.

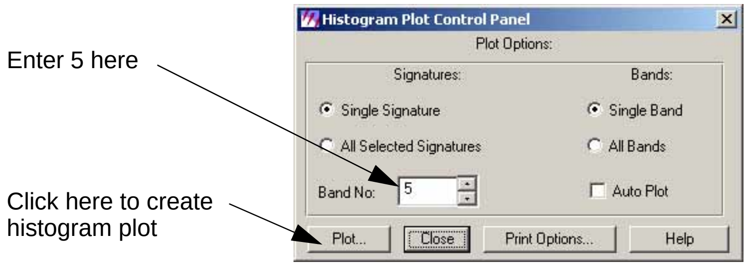
Plot Histograms

The Histogram Plot Control Panel allows you to analyze the histograms for the layers to make your own evaluations and comparisons. A histogram can be created with one or more signatures. If you create a histogram for a single signature, then the active signature, which is next to the **>**, is used. If you create a histogram for multiple signatures, then the selected signatures are used.

1. In the Signature Editor, move the **>** prompt to the signature for **Agricultural Field_1** by clicking under the **>** column.

2. In the Signature Editor menu bar, select **View | Histograms** or click on the Histogram icon .

   The Histogram Plot Control Panel and the Histogram dialogs open.


Histogram Plot Control Panel

Plot Options:

Signatures:

● Single Signature

○ All Selected Signatures

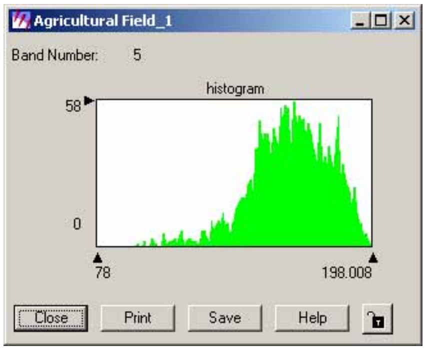
Band No: 5

Bands:

● Single Band

○ All Bands

☐ Auto Plot


Agricultural Field_1

Band Number: 5

3. In the Histogram Plot Control Panel dialog, change the **Band No** number field to **5** in order to view the histogram for band 5 (i.e., layer 5).

4. Click **Plot**.

   The Histogram dialog changes to display the histogram for band 5. You can change the different plot options and select different signatures to see the differences in histograms for various signatures and bands.

5. In the Histogram Plot Control Panel dialog, click **Close**. The two Histogram dialogs close.

Compute Signature Separability

The Signature Separability utility computes the statistical distance between signatures. This distance can be used to determine how distinct your signatures are from one another. This utility can also be used to determine the best subset of layers to use in the classification.

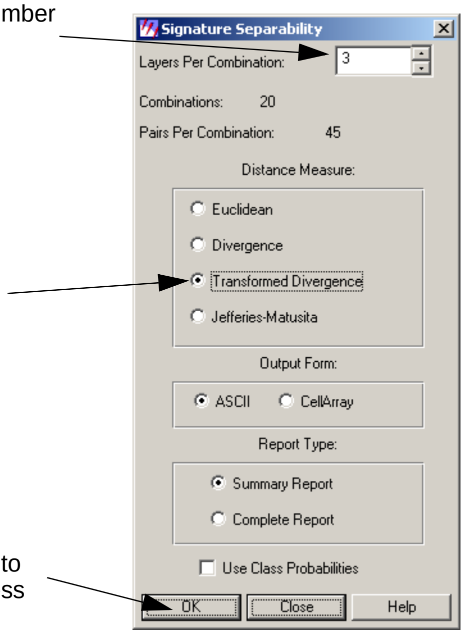
The distances are based on the following formulas:

- euclidean spectral distances between their means
- Jeffries-Matusita distance
- divergence
- transformed divergence

The Signature Separability utility can be performed with multiple signatures. If you do not have any signatures selected, then all of the parametric signatures are used.

1. In the Signature Editor, select all of the parametric signatures.
2. In the Signature Editor menu bar, select **Evaluate | Separability**.

   The Signature Separability dialog opens.



Set this number field to 3

Click here

Click here to start process

Layers Per Combination: 3

Combinations: 20

Pairs Per Combination: 45

Distance Measure: ● Transformed Divergence

Output Form: ● ASCII

Report Type: ● Summary Report

☐ Use Class Probabilities

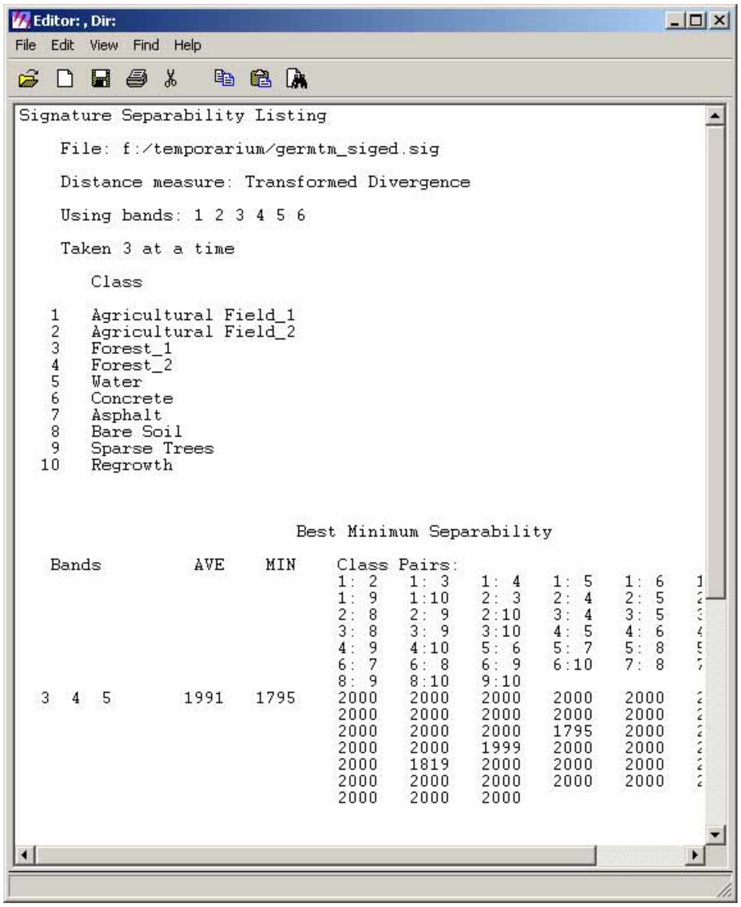
Signature Separability
Distance Measure:
○ Euclidean
○ Divergence
● Transformed Divergence
○ Jefferies-Matusita
Output Form:
● ASCII
○ CellArray
Report Type:
● Summary Report
○ Complete Report
OK
Close
Help

3. In the Signature Separability dialog, set the **Layers Per Combination** number field to **3**, so that three layers are used for each combination.
4. Click **Transformed Divergence** under **Distance Measure** to use the divergence algorithm for calculating the separability.
5. Confirm that the **Summary Report** checkbox is turned on under **Report Type**, in order to output a summary of the report.

The summary lists the separability listings for only those band combinations with best average and best minimum separability.

6. In the Signature Separability dialog, click **OK** to begin the process.

   When the process is complete, the IMAGINE Text Editor opens, displaying the report.



```
Signature Separability Listing

    File: f:/temporarium/germtm_siged.sig

    Distance measure: Transformed Divergence

    Using bands: 1 2 3 4 5 6

    Taken 3 at a time

       Class

   1   Agricultural Field_1
   2   Agricultural Field_2
   3   Forest_1
   4   Forest_2
   5   Water
   6   Concrete
   7   Asphalt
   8   Bare Soil
   9   Sparse Trees
  10   Regrowth



                    Best Minimum Separability

   Bands         AVE    MIN    Class Pairs:
                               1: 2   1: 3   1: 4   1: 5   1: 6   1
                               1: 9   1:10   2: 3   2: 4   2: 5
                               2: 8   2: 9   2:10   3: 4   3: 5
                               3: 8   3: 9   3:10   4: 5   4: 6
                               4: 9   4:10   5: 6   5: 7   5: 8
                               6: 7   6: 8   6: 9   6:10   7: 8
                               8: 9   8:10   9:10
  3  4  5       1991   1795    2000   2000   2000   2000   2000
                               2000   2000   2000   2000   2000
                               2000   2000   2000   1795   2000
                               2000   2000   1999   2000   2000
                               2000   1819   2000   2000   2000
                               2000   2000   2000   2000   2000
                               2000   2000   2000
```

This report shows that layers (i.e., bands) 2, 4, and 5 are the best layers to use for identifying features.

7. In the Text Editor menu bar, select **File | Close** to close the Editor.

8. In the Signature Separability dialog, click **Close**.

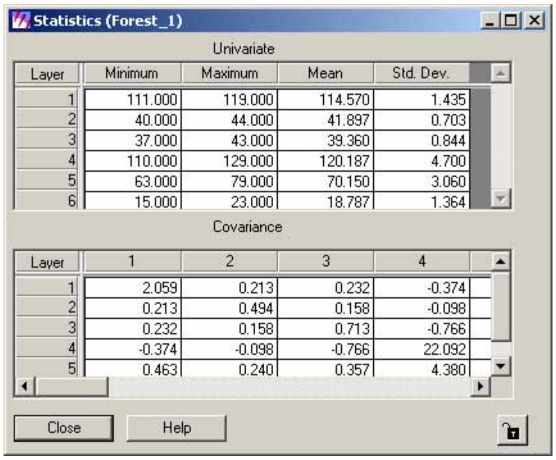
Check Statistics

The Statistics utility allows you to analyze the statistics for the layers to make your own evaluations and comparisons. Statistics can be generated for one signature at a time. The active signature, which is next to the **>**, is used.

1. In the Signature Editor, move the **>** prompt to the signature for **Forest_1**.

2. In the Signature Editor menu bar, select **View | Statistics** or click on the Statistics icon Σ.

The Statistics dialog opens.



Univariate

Layer	Minimum	Maximum	Mean	Std. Dev.
1	111.000	119.000	114.570	1.435
2	40.000	44.000	41.897	0.703
3	37.000	43.000	39.360	0.844
4	110.000	129.000	120.187	4.700
5	63.000	79.000	70.150	3.060
6	15.000	23.000	18.787	1.364

Covariance

Layer	1	2	3	4
1	2.059	0.213	0.232	-0.374
2	0.213	0.494	0.158	-0.098
3	0.232	0.158	0.713	-0.766
4	-0.374	-0.098	-0.766	22.092
5	0.463	0.240	0.357	4.380

3. After viewing the information in the Statistics dialog, click **Close**.

Perform Supervised Classification

The decision rules for the supervised classification process are multilevel:

- nonparametric
- parametric

In this example, use both nonparametric and parametric decision rules.

See "Chapter 6: Classification" *of the* ERDAS Field Guide *for more information on decision rules.*

Nonparametric

If the signature is nonparametric (i.e., Feature Space AOI), then the following decision rules are offered:

- feature space
- parallelepiped

With nonparametric signatures you must also decide the overlap rule and the unclassified rule.

NOTE: All signatures have a nonparametric definition, due to their parallelepiped boundaries.

Parametric

For parametric signatures, the following decision rules are provided:

- maximum likelihood
- Mahalanobis distance
- minimum distance

In this tour guide, use the maximum likelihood decision rule.

Output File

The Supervised Classification utility outputs a thematic raster layer (.img extension) and/or a distance file (.img extension). The distance file can be used for post-classification thresholding. The thematic raster layer automatically contains the following data:

- class values
- class names
- color table
- statistics
- histogram

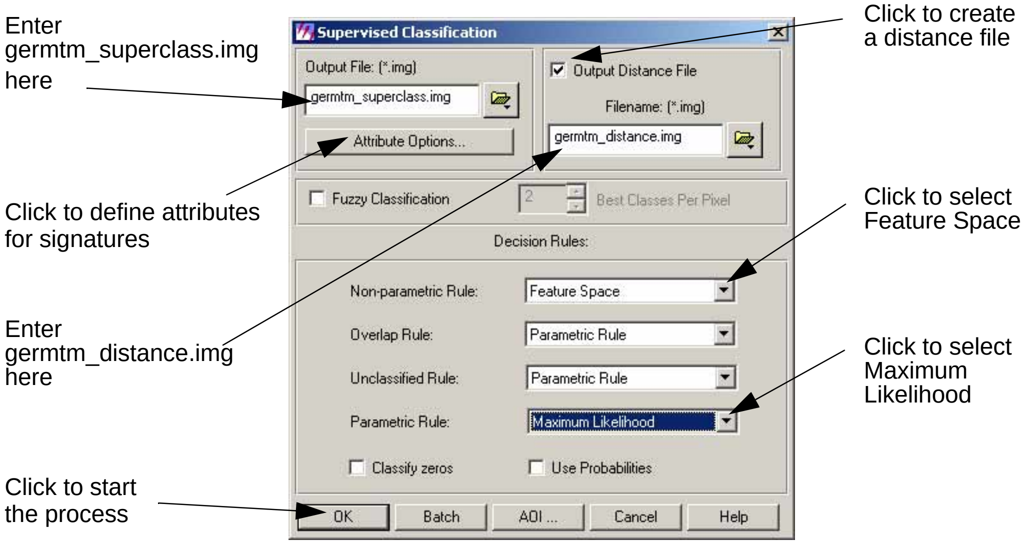
The image file also contains any signature attributes that were selected in the Supervised Classification utility.

1. In the Signature Editor, select all of the signatures so that they are all used in the classification process (if none of the signatures are selected, then they are all used by default.)

2. In the Signature Editor menu bar, select **Classify | Supervised** to perform a supervised classification.

   NOTE: You may also access the Supervised Classification utility from the Classification dialog.

   The Supervised Classification dialog opens.



Enter germtm_superclass.img here

Click to create a distance file

Click to define attributes for signatures

Click to select Feature Space

Enter germtm_distance.img here

Click to select Maximum Likelihood

Click to start the process

Supervised Classification

Output File: (*.img) germtm_superclass.img

☑ Output Distance File

Filename: (*.img) germtm_distance.img

☐ Fuzzy Classification 2 Best Classes Per Pixel

Decision Rules:

Non-parametric Rule: Feature Space

Overlap Rule: Parametric Rule

Unclassified Rule: Parametric Rule

Parametric Rule: Maximum Likelihood

☐ Classify zeros ☐ Use Probabilities

3. In the Supervised Classification dialog, under **Output File**, type in **germtm_superclass.img**.

   This is the name for the thematic raster layer.

4. Click the **Output Distance File** checkbox to activate it.

   In this example, you are creating a distance file that can be used to threshold the classified image file.

5. Under **Filename**, enter **germtm_distance.img** in the directory of your choice.

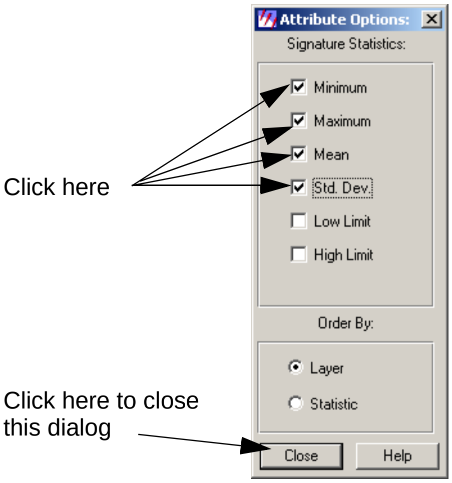
   This is the name for the distance image file.

   NOTE: Make sure you remember the directory in which the output file is saved. It is important when you are trying to display the output file in a Viewer.

Select Attribute Options

1. In the Supervised Classification dialog, click **Attribute Options**.

   The Attribute Options dialog opens.



Attribute Options:

Signature Statistics:

☑ Minimum
☑ Maximum
☑ Mean
☑ Std. Dev.
☐ Low Limit
☐ High Limit

Order By:

● Layer
○ Statistic

Click here

Click here to close this dialog

The Attribute Options dialog allows you to specify the statistical information for the signatures that you want to be in the output classified layer. The statistics are based on the data file values for each layer for the signatures, not the entire classified image file. This information is located in the Raster Attribute Editor.

2. In the Attribute Options dialog, click **Minimum**, **Maximum**, **Mean**, and **Std. Dev.**, so that the signatures in the output thematic raster layer have this statistical information.

3. Confirm that the **Layer** checkbox is turned on, so that the information is presented in the Raster Attribute Editor by layer.

4. In the Attribute Options dialog, click **Close** to remove this dialog.

Classify the Image

1. In the Supervised Classification dialog, click on the **Non-parametric Rule** popup list to select **Feature Space**.

   You do not need to use the **Classify Zeros** option here because there are no background zero data file values in the **germtm.img** file.

2. Click **OK** in the Supervised Classification dialog to classify the **germtm.img** file using the signatures in the Signature Editor.

   A Job Status dialog displays, indicating the progress of the function.

3. When the process is 100% complete, click **OK** in the Job Status dialog.

See "Chapter 6: Classification" *of the* ERDAS Field Guide *for information on how the pixels are classified.*

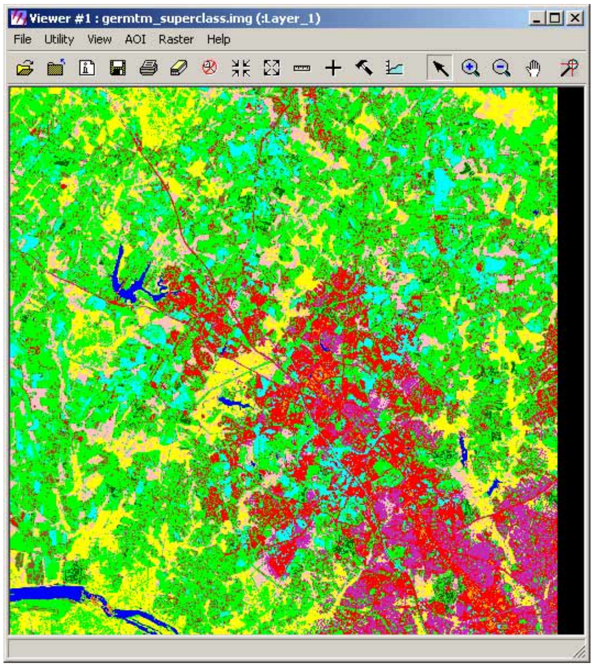
4. Select **File | Close** from the Signature Editor menu bar and from the Viewer #2 menu bar. Click **Yes** when asked if you would like to save the changes to the Signature Editor.

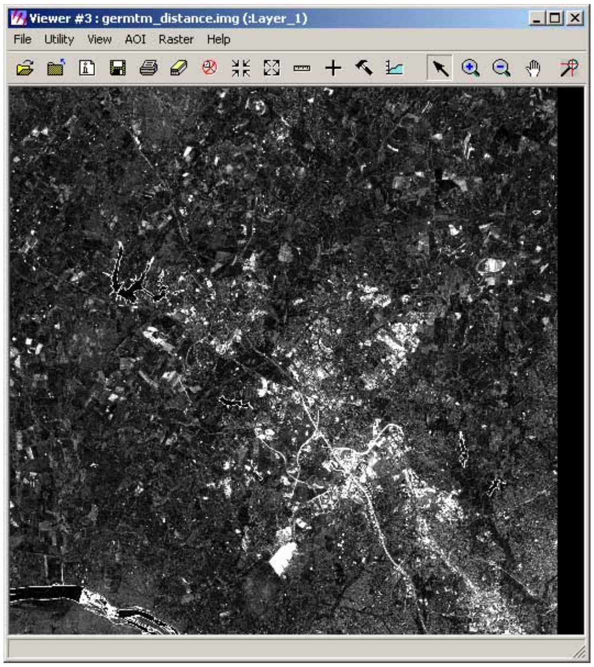
5. Click **Close** in the Raster tool palette.

6. Select **File | Clear** from the Viewer #1 menu bar.

7. Proceed to the:
    - “Perform Unsupervised Classification” section on page 483 to classify the same image using the ISODATA algorithm.
    - “Evaluate Classification” section on page 486 to analyze the classes and test the accuracy of the classification, or

The super classification image is pictured on the left, and the distance image is pictured on the right.


Viewer #1 : germtm_superclass.img (:Layer_1)
File Utility View AOI Raster Help


Viewer #3 : germtm_distance.img (:Layer_1)
File Utility View AOI Raster Help

Perform Unsupervised Classification

This section shows you how to create a thematic raster layer by letting the software identify statistical patterns in the data without using any ground truth data.

ERDAS IMAGINE uses the ISODATA algorithm to perform an unsupervised classification. The ISODATA clustering method uses the minimum spectral distance formula to form clusters. It begins with either arbitrary cluster means or means of an existing signature set, and each time the clustering repeats, the means of these clusters are shifted. The new cluster means are used for the next iteration.

The ISODATA utility repeats the clustering of the image until either a maximum number of iterations has been performed, or a maximum percentage of unchanged pixel assignments has been reached between two iterations.

Performing an unsupervised classification is simpler than a supervised classification, because the signatures are automatically generated by the ISODATA algorithm.

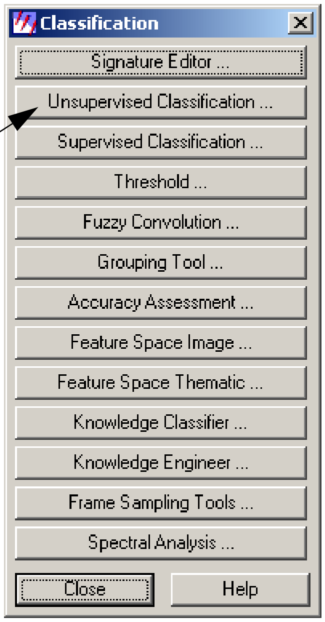
In this example, you generate a thematic raster layer using the ISODATA algorithm.

Preparation

You must have ERDAS IMAGINE running.

1. Click on the Classifier icon Classifier in the ERDAS IMAGINE icon panel to start the Classification utility.

The **Classification** menu opens.

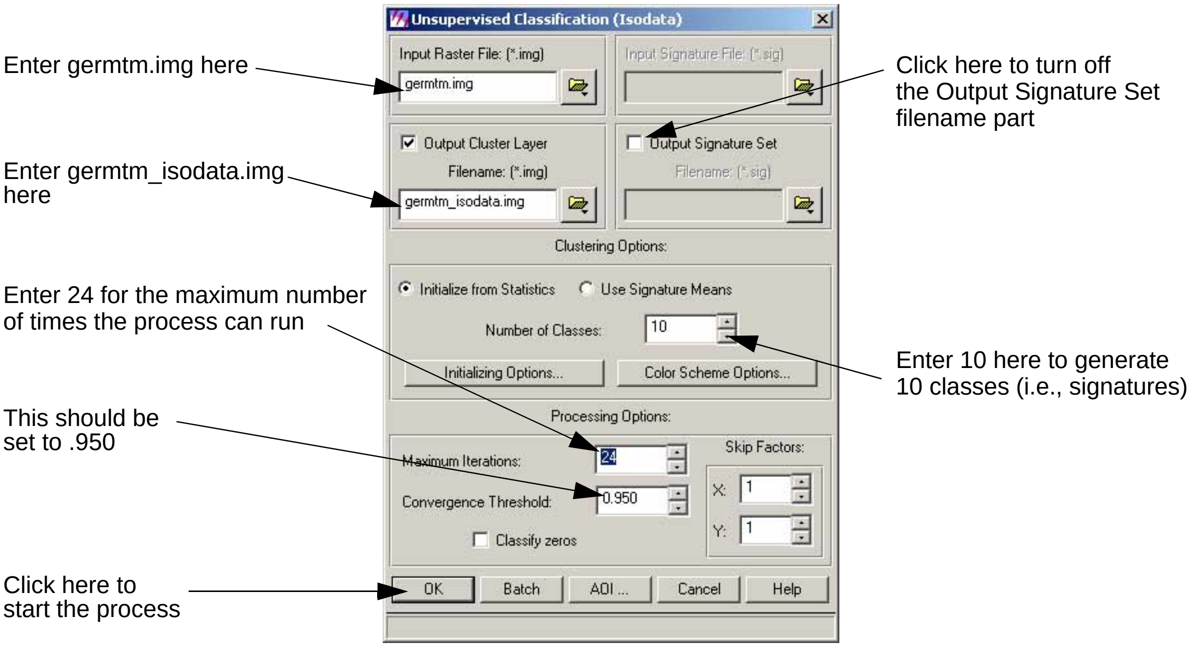

Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Grouping Tool ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Frame Sampling Tools ...
Spectral Analysis ...
Close
Help

Click here to start the Unsupervised Classification utility

Generate Thematic Raster Layer

1. Select **Unsupervised Classification** from the **Classification** menu to perform an unsupervised classification using the ISODATA algorithm.

The Unsupervised Classification dialog opens.



Unsupervised Classification (Isodata)

Input Raster File: (*.img) germtm.img

Input Signature File: (*.sig)

☑ Output Cluster Layer

Filename: (*.img) germtm_isodata.img

☐ Output Signature Set

Filename: (*.sig)

Clustering Options:

● Initialize from Statistics ○ Use Signature Means

Number of Classes: 10

Processing Options:

Maximum Iterations: 24

Convergence Threshold: 0.950

Skip Factors: X: 1 Y: 1

☐ Classify zeros

Enter germtm.img here

Enter germtm_isodata.img here

Enter 24 for the maximum number of times the process can run

This should be set to .950

Click here to start the process

Click here to turn off the Output Signature Set filename part

Enter 10 here to generate 10 classes (i.e., signatures)

2. Click **Close** in the **Classification** menu to clear it from the screen.

3. In the Unsupervised Classification dialog under **Input Raster File**, enter **germtm.img**. This is the image file that you are going to classify.

4. Under **Output Cluster Layer**, enter **germtm_isodata.img** in the directory of your choice.

   This is the name for the output thematic raster layer.

5. Click **Output Signature Set** to turn off the checkbox.

   For this example, do not create a signature set. The **Output Signature Set** filename part is disabled.

Set Initial Cluster Options

The **Clustering Options** allow you to define how the initial clusters are generated.

1. Confirm that the **Initialize from Statistics** checkbox under **Clustering Options** is turned on.

   This generates arbitrary clusters from the file statistics for the image file.

2. Enter a **10** in the **Number of Classes** number field.

Set Processing Options

The Processing Options allow you to specify how the process is performed.

1. Enter **24** in the **Maximum Iterations** number field under **Processing Options**.

This is the maximum number of times that the ISODATA utility reclusters the data. It prevents this utility from running too long, or from potentially getting stuck in a cycle without reaching the convergence threshold.

2. Confirm that the **Convergence Threshold** number field is set to **.95**.

Convergence Threshold

The convergence threshold is the maximum percentage of pixels that has cluster assignments that can go unchanged between iterations. This threshold prevents the ISODATA utility from running indefinitely.

By specifying a convergence threshold of .95, you are specifying that as soon as 95% or more of the pixels stay in the same cluster between one iteration and the next, the utility should stop processing. In other words, as soon as 5% or fewer of the pixels change clusters between iterations, the utility stops processing.

3. Click **OK** in the Unsupervised Classification dialog to start the classification process. The Unsupervised Classification dialog closes automatically.

   A Job Status dialog displays, indicating the progress of the function.

4. Click **OK** in the Job Status dialog when the process is 100% complete.
5. Proceed to the Evaluate Classification section to analyze the classes so that you can identify and assign class names and colors.

Evaluate Classification

After a classification is performed, the following methods are available for evaluating and testing the accuracy of the classification:

- classification overlay
- thresholding
- recode classes
- accuracy assessment

See "Chapter 6: Classification" *of the* ERDAS Field Guide *for information on accuracy assessment.*

Create Classification Overlay

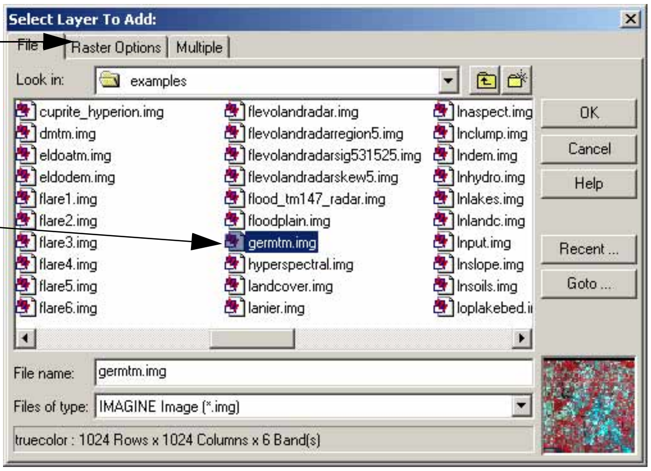
In this example, use the Raster Attribute Editor to compare the original image data with the individual classes of the thematic raster layer that was created from the unsupervised classification (**germtm_isodata.img**). This process helps identify the classes in the thematic raster layer. You may also use this process to evaluate the classes of a thematic layer that was generated from a supervised classification.

Preparation

ERDAS IMAGINE should be running and you should have a Viewer open.

1. Select **File | Open | Raster Layer** from the Viewer menu bar or click the Open icon in the tool bar to display the **germtm.img** continuous raster layer.

   The Select Layer To Add dialog opens.



Click on this file tab to see the raster options

Click here to select germtm.img

Look in: examples

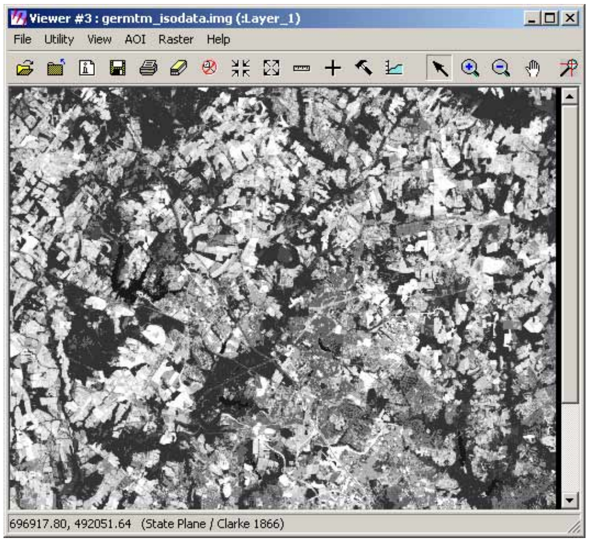
File name: germtm.img

Files of type: IMAGINE Image (*.img)

truecolor : 1024 Rows x 1024 Columns x 6 Band(s)

Select Layer To Add:
File
Raster Options
Multiple
cuprite_hyperion.img
dmtm.img
eldoatm.img
eldodem.img
flare1.img
flare2.img
flare3.img
flare4.img
flare5.img
flare6.img
flevolandradar.img
flevolandradarregion5.img
flevolandradarsig531525.img
flevolandradarskew5.img
flood_tm147_radar.img
floodplain.img
germtm.img
hyperspectral.img
landcover.img
lanier.img
lnaspect.img
lnclump.img
lndem.img
lnhydro.img
lnlakes.img
lnlandc.img
lnput.img
lnslope.img
lnsoils.img
loplakebed.i
OK
Cancel
Help
Recent ...
Goto ...

2. In the Select Layer To Add dialog under **Filename**, select **germtm.img**.
3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
4. Set **Layers to Colors** at **4**, **5**, and **3**, respectively.
5. Click **OK** in the Select Layer To Add dialog to display the image file.



696917.80, 492051.64 (State Plane / Clarke 1866)

Viewer #3 : germtm_isodata.img (:Layer_1)
File
Utility
View
AOI
Raster
Help

6. Click the Open icon again in the Viewer tool bar to display the thematic raster layer, **germtm_isodata.img**, over the **germtm.img** file.

The Select Layer To Add dialog reopens.

7. Under **Filename**, open the directory in which you previously saved **germtm_isodata.img** by entering the directory path name in the text entry field and pressing the Return key on your keyboard.

   You are going to evaluate/identify the classes in this file.

8. Select the file **germtm_isodata.img** from the list of files in the directory you just opened.
9. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
10. Click **Clear Display** to turn off this checkbox.
11. Click **OK** in the Select Layer To Add dialog to display the image file.

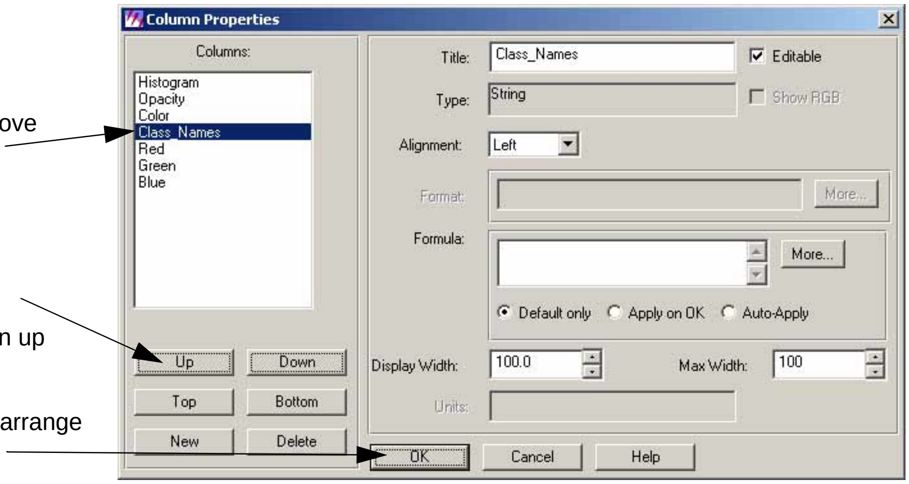
Open Raster Attribute Editor

1. Select **Raster | Attributes** from the Viewer menu bar.

   The Raster Attribute Editor displays.

2. In the Raster Attribute Editor, select **Edit | Column Properties** to rearrange the columns in the CellArray so that they are easier to view.

   The Column Properties dialog opens.


Column Properties
Columns:
Histogram
Opacity
Color
Class_Names
Red
Green
Blue
Up
Down
Top
Bottom
New
Delete
Format:
More...
Formula:
More...
Show RGB
Units:
OK
Cancel
Help

Title: Class_Names ☑ Editable

Type: String ☐ Show RGB

Alignment: Left

● Default only ○ Apply on OK ○ Auto-Apply

Display Width: 100.0 Max Width: 100

Click here to move this column

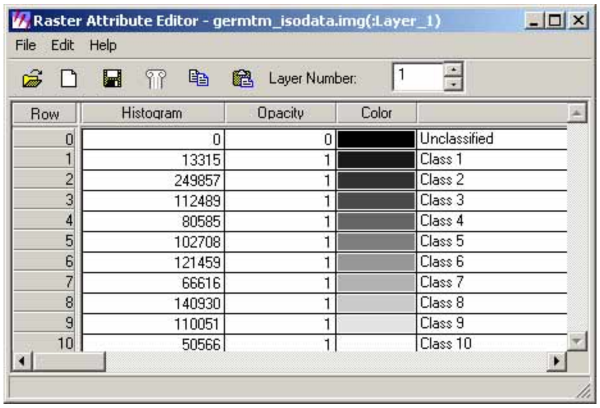
Click here to move the selected column up

Click here to rearrange the columns

3. In the Column Properties dialog under **Columns**, select **Opacity**, then click **Up** to move **Opacity** so that it is under **Histogram**.
4. Select **Class_Names**, then click **Up** to move **Class_Names** so that it is under **Color**.
5. Click **OK** in the Column Properties dialog to rearrange the columns in the Raster Attribute Editor.

The Column Properties dialog closes.

The data in the Raster Attribute Editor CellArray should appear similar to the following example:


Raster Attribute Editor - germtm_isodata.img(:Layer_1)
File
Edit
Help
Layer Number:
1

Row	Histogram	Opacity	Color	
0	0	0	■	Unclassified
1	13315	1	■	Class 1
2	249857	1	■	Class 2
3	112489	1	■	Class 3
4	80585	1	■	Class 4
5	102708	1	■	Class 5
6	121459	1	■	Class 6
7	66616	1	■	Class 7
8	140930	1	■	Class 8
9	110051	1	■	Class 9
10	50566	1	■	Class 10

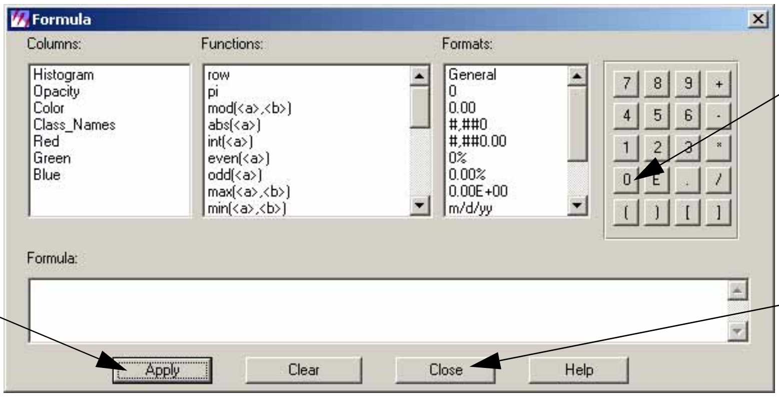
Analyze Individual Classes

Before you can begin to analyze the classes individually, you need to set the opacity for all of the classes to zero.

1. In the Raster Attribute Editor, click on the word **Opacity** at the top of the **Opacity** column to select all of the classes.

2. In the Raster Attribute Editor, click on the word **Opacity** at the top of the **Opacity** column to select the Opacity column. Right-hold on the word Opacity and select **Formula** from the **Column Options** menu.

   The Formula dialog opens.


Formula
Columns:
Histogram
Opacity
Color
Class_Names
Red
Green
Blue
Functions:
row
pi
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
max(<a>,<b>)
min(<a>,<b>)
Formats:
General
0
0.00
#,##0
#,##0.00
0%
0.00%
0.00E+00
m/d/yy
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
Formula:
Apply
Clear
Close
Help

Click here to enter a 0 in the Formula

Click here to apply a 0 value to the Opacity column

Click here to close this dialog

3. In the Formula dialog, click **0** in the number pad.

   A **0** is placed in the **Formula** field.

4. In the Formula dialog, click **Apply** to change all of the values in the **Opacity** column to **0**, and then click **Close**.

5. In the Raster Attribute Editor, hold on the color patch under **Color** for **Class 1** in the CellArray and change the color to **Yellow**. This provides better visibility in the Viewer.

6. Change the **Opacity** for **Class 1** in the CellArray to **1** and then press Return on the keyboard. This class is shown in the Viewer.

7. In the Viewer menu bar, select **Utility | Flicker** to analyze which pixels have been assigned to this class.

   The Viewer Flicker dialog opens.

8. Turn on the **Auto Mode** in the Viewer Flicker dialog.

   The flashing black pixels in the **germtm.img** file are the pixels of this class. These areas are water.

9. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 1**. Change this name to **Water** and then press the Return key on the keyboard.

10. In the Raster Attribute Editor, hold on the **Color** patch for **Water**. Select **Blue** from the popup list.

11. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Water** back to **0**. Press the Return key on the keyboard.

12. Change the **Color** for **Class 2** in the CellArray to **Yellow** for better visibility in the Viewer.

13. Change the **Opacity** for **Class 2** to **1** and press the Return key on the keyboard. This class is shown in the Viewer.

Use the Flicker Utility

1. In the Viewer menu bar, select **Utility | Flicker** to analyze which pixels were assigned to this class.

   The Viewer Flicker dialog opens.

2. Turn on the **Auto Mode** in the Viewer Flicker dialog.

   The flashing red pixels in the **germtm.img** file should be the pixels of this class. These are forest areas.

3. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 2**. (You may need to double-click in the column.) Change this name to **Forest**, then press the Return key on the keyboard.

4. In the Raster Attribute Editor, hold on the **Color** patch for **Forest** and select **Pink** from the popup list.

5. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Forest** back to **0**. Press the Return key on the keyboard.

6. Repeat these steps with each class so that you can see how the pixels are assigned to each class. You may also try selecting more than one class at a time.

7. Continue assigning names and colors for the remaining classes in the Raster Attribute Editor CellArray.

8. In the Raster Attribute Editor, select **File | Save** to save the data in the CellArray.

9. Select **File | Close** from the Raster Attribute Editor menu bar.

10. Select **File | Clear** from the Viewer menu bar.

Use Thresholding

The Thresholding utility allows you to refine a classification that was performed using the Supervised Classification utility. The Thresholding utility determines which pixels in the new thematic raster layer are most likely to be incorrectly classified.

This utility allows you to set a distance threshold for each class in order to screen out the pixels that most likely do not belong to that class. For all pixels that have distance file values greater than a threshold you set, the class value in the thematic raster layer is set to another value.

The threshold can be set:

- with numeric input, using chi-square statistics, confidence level, or Euclidean spectral distance, or
- interactively, by viewing the histogram of one class in the distance file while using the mouse to specify the threshold on the histogram graph.

Since the chi-square table is built-in, you can enter the threshold value in the confidence level unit and the chi-square value is automatically computed.

In this example, you threshold the output thematic raster layer from the supervised classification (**germtm_superclass.img**).

Preparation

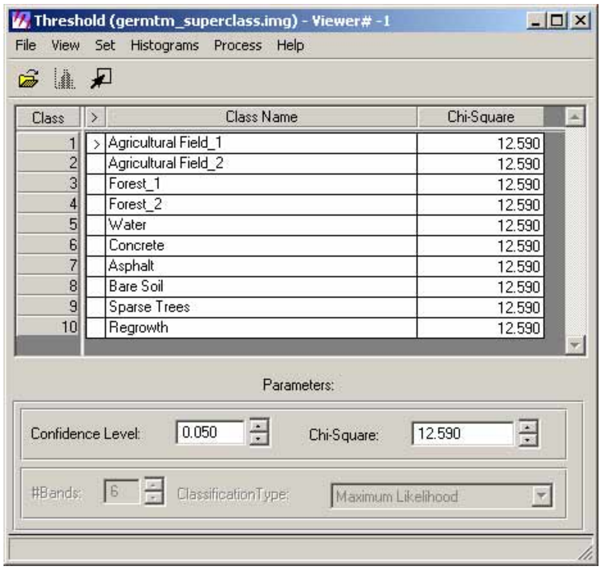
ERDAS IMAGINE must be running and you must have **germtm_superclass.img** displayed in a Viewer.

1. Click on the Classifier icon Classifier in the ERDAS IMAGINE icon panel to start the Classification utility.

   The **Classification** menu displays.

2. Select **Threshold** from the **Classification** menu to start the Threshold dialog.

   The Threshold dialog opens.


Threshold (germtm_superclass.img) - Viewer# -1
File View Set Histograms Process Help
Parameters:

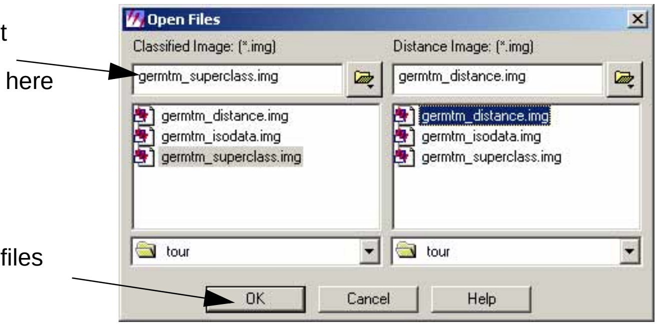
Class	>	Class Name	Chi-Square
1	>	Agricultural Field_1	12.590
2		Agricultural Field_2	12.590
3		Forest_1	12.590
4		Forest_2	12.590
5		Water	12.590
6		Concrete	12.590
7		Asphalt	12.590
8		Bare Soil	12.590
9		Sparse Trees	12.590
10		Regrowth	12.590

Confidence Level: 0.050
Chi-Square: 12.590
#Bands: 6
ClassificationType: Maximum Likelihood

3. Click **Close** in the **Classification** menu to clear it from the screen.

4. In the Threshold dialog, select **File | Open** or click the Open icon to define the classified image and distance image files.

   The Open Files dialog opens.



Classified Image: (*.img) germtm_superclass.img
Distance Image: (*.img) germtm_distance.img

Classified Image files:
germtm_distance.img
germtm_isodata.img
germtm_superclass.img

Distance Image files:
germtm_distance.img
germtm_isodata.img
germtm_superclass.img

Directory: tour
Directory: tour

Type in the correct path name and then press Return here

Click here to load files

Select Classified and Distance Images

1. In the Open Files dialog under **Classified Image**, open the directory in which you previously saved **germtm_superclass.img** by entering the directory path name in the text window and pressing Return on your keyboard.

2. Select the file **germtm_superclass.img** from the list of files in the directory you just opened.

   This is the classified image file that is going to be thresholded.

3. In the Open Files dialog, under **Distance Image**, open the directory in which you previously saved **germtm_distance.img** by entering the directory path name in the text entry field and pressing Return on your keyboard.

4. Select the file **germtm_distance.img** from the list of files in the directory you just opened.

   This is the distance image that was created when the **germtm_superclass.img** file was created. A distance image file for the classified image is necessary for thresholding.

5. Click **OK** in the Open Files dialog to load the files.

6. In the Threshold dialog, select **View | Select Viewer** and then click in the Viewer that is displaying the **germtm_superclass.img** file.

Compute and Evaluate Histograms

1. In the Threshold dialog, select **Histograms | Compute**.

   The histograms for the distance image file are computed. There is a separate histogram for each class in the classified image file.

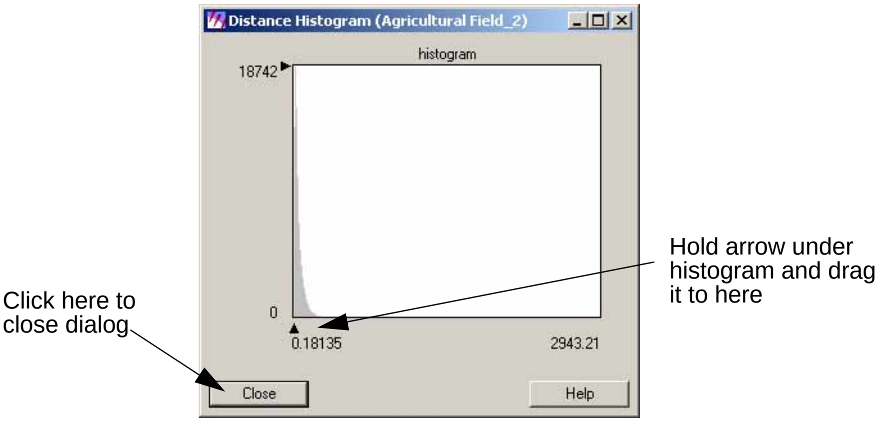
   The Job Status dialog opens as the histograms are computed. This dialog automatically closes when the process is completed.

2. If desired, select **Histograms | Save** to save this histogram file.

3. In the CellArray of the Threshold dialog, move the **>** prompt to the **Agricultural Field_2** class by clicking under the **>** column in the cell for **Class 2**.

4. Select **Histograms | View**.

   The Distance Histogram for **Agricultural Field_2** displays.



Click here to close dialog

Hold arrow under histogram and drag it to here

5. Select the arrow on the X axis of the histogram graph to move it to the position where you want to threshold the histogram.

The **Chi-Square** value in the Threshold dialog is updated for the current class (**Agricultural Field_2**) as you move the arrow.

6. In the Threshold dialog CellArray, move the **>** prompt to the next class.

   The histogram is updated for this class.

7. Repeat the steps, thresholding the histogram for each class in the Threshold dialog CellArray.

See "Chapter 6: Classification" *of the* ERDAS Field Guide *for information on thresholding.*

8. After you have thresholded the histogram for each class, click **Close** in the Distance Histogram dialog.

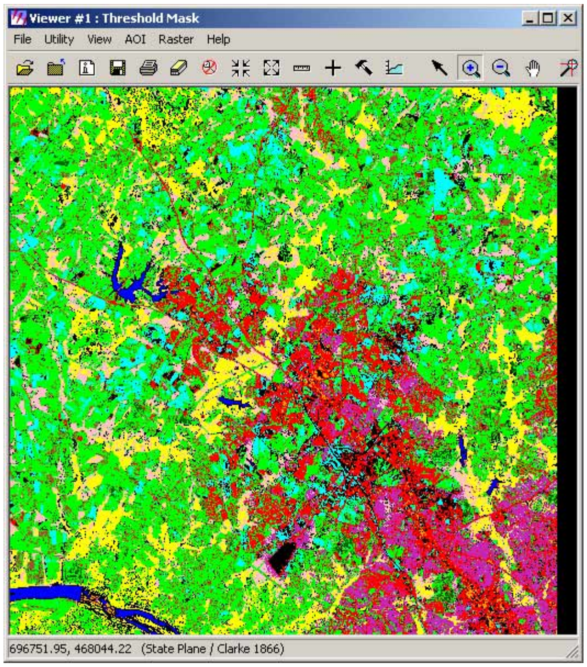
Apply Colors

1. In the Threshold dialog, select **View | View Colors | Default Colors**.

   Use the default setting so that the thresholded pixels appear black and those pixels remaining appear in their classified color in the thresholded image.

2. In the Threshold dialog, select **Process | To Viewer**.

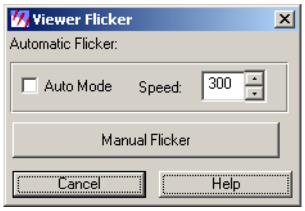
   The thresholded image is placed in the Viewer over the **germtm_superclass.img** file. Yours likely looks different from the one pictured here.


Viewer #1 : Threshold Mask
File Utility View AOI Raster Help
696751.95, 468044.22 (State Plane / Clarke 1866)

Use the Flicker Utility

1. In the Viewer menu bar, select **Utility | Flicker** to see how the classes were thresholded.

   The Viewer Flicker dialog opens.



2. When you are finished observing the thresholding, click **Cancel** in the Viewer Flicker dialog.
3. In the Viewer, select **View | Arrange Layers**.

   The Arrange Layers dialog opens.
4. In the Arrange Layers dialog, right-hold over the thresholded layer (**Threshold Mask**) and select **Delete Layer** from the **Layer Options** menu.
5. Click **Apply** and then **Close** in the Arrange Layers dialog. When asked if you would like to save your changes, click **No**.
6. In the Threshold dialog, select **Process | To File**.

   The Threshold to File dialog opens.

Process Threshold

1. In the Threshold to File dialog under **Output Image**, enter the name **germtm_thresh.img** in the directory of your choice.

   This is the file name for the thresholded image.
2. Click **OK** to output the thresholded image to a file.

   The Threshold to File dialog closes.
3. Wait for the thresholding process to complete, and then select **File | Close** from the Threshold dialog menu bar.
4. Select **File | Clear** from the Viewer menu bar.

   NOTE: The output file that is generated by thresholding a classified image can be further analyzed and modified in various ERDAS IMAGINE utilities, including the Image Interpreter, Raster Attribute Editor, and Spatial Modeler.

Recode Classes

After you analyze the pixels, you may want to recode the thematic raster layer to assign a new class value number to any or all classes, creating a new thematic raster layer using the new class numbers. You can also combine classes by recoding more than one class to the same new class number. Use the **Recode** function under **Interpreter** (icon) **| GIS Analysis** to recode a thematic raster layer.

See "Chapter 11: Geographic Information Systems" *of the* ERDAS Field Guide *for more information on recoding.*

Use Accuracy Assessment

The Accuracy Assessment utility allows you to compare certain pixels in your thematic raster layer to reference pixels, for which the class is known. This is an organized way of comparing your classification with ground truth data, previously tested maps, aerial photos, or other data.

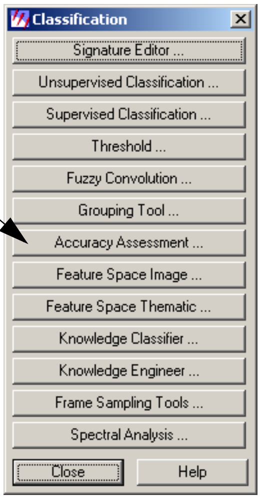
In this example, you perform an accuracy assessment using the output thematic raster layer from the supervised classification (**germtm_superclass.img**).

Preparation

ERDAS IMAGINE must be running and you must have **germtm.img** displayed in a Viewer.

1. Click the Classifier icon Classifier in the ERDAS IMAGINE icon panel.

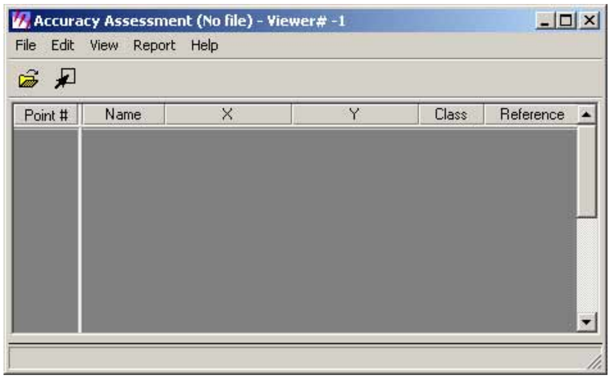
   The **Classification** menu displays.


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Grouping Tool ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Frame Sampling Tools ...
Spectral Analysis ...
Close
Help

Click here to start the Accuracy Assessment Utility.

2. Select **Accuracy Assessment** from the **Classification** menu to start the Accuracy Assessment utility.

   The Accuracy Assessment dialog opens.


Accuracy Assessment (No file) - Viewer# -1
File
Edit
View
Report
Help
Point #
Name
X
Y
Class
Reference

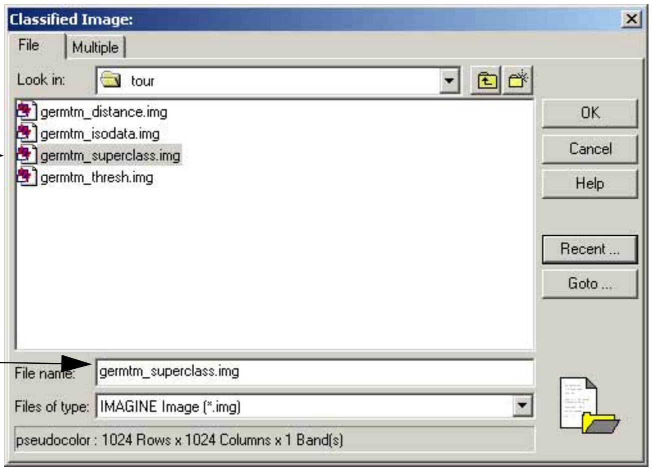
Check the Accuracy Assessment CellArray

The Accuracy Assessment CellArray contains a list of class values for the pixels in the classified image file and the class values for the corresponding reference pixels. The class values for the reference pixels are input by you. The CellArray data reside in the classified image file (e.g., **germtm_superclass.img**).

1. Click **Close** in the **Classification** menu to clear it from the screen.

2. In the Accuracy Assessment dialog, select **File | Open** or click the Open icon .

   The Classified Image dialog opens.


Classified Image:
File
Multiple
OK
Cancel
Help
Recent ...
Goto ...

Look in: tour

germtm_distance.img
germtm_isodata.img
germtm_superclass.img
germtm_thresh.img

File name: germtm_superclass.img

Files of type: IMAGINE Image (*.img)

pseudocolor : 1024 Rows x 1024 Columns x 1 Band(s)

Select germtm_superclass.img from this list

Enter the correct path name here

3. In the Classified Image dialog, under **Filename**, open the directory in which you previously saved **germtm_superclass.img** by entering the directory path name in the text entry field and pressing Return on your keyboard.

4. Select the file **germtm_superclass.img** from the list of files in the directory you just opened.

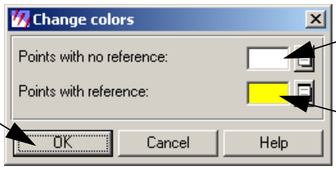
   This is the classified image file that is used in the accuracy assessment.

5. Click **OK** in the Classified Image dialog to load the file.

6. In the Accuracy Assessment dialog, select **View | Select Viewer** or click the Select Viewer icon , then click in the Viewer that is displaying the **germtm.img** file.

7. In the Accuracy Assessment dialog, select **View | Change Colors**.

   The Change colors dialog opens.



Change colors

Points with no reference: ⬜

Points with reference: 🟨

Click here

This color should be set to white

This color should be set to yellow

In the Change colors dialog, the **Points with no reference** color patch should be set to **White**. These are the random points that have not been assigned a reference class value.

The **Points with reference** color patch should be set to **Yellow**. These are the random points that have been assigned a reference class value.

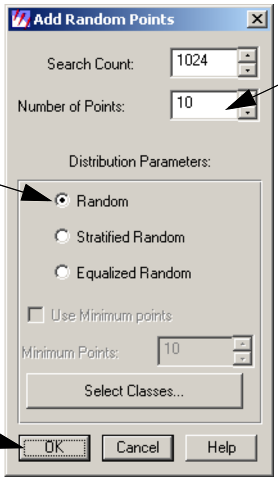
8. Click **OK** in the Change colors dialog to accept the default colors.

Generate Random Points

The Add Random Points utility generates random points throughout your classified image. After the points are generated, you must enter the class values for these points, which are the reference points. These reference values are compared to the class values of the classified image.

1. In the Accuracy Assessment dialog, select **Edit | Create/Add Random Points**.

   The Add Random Points dialog opens.



Add Random Points

Search Count: 1024

Number of Points: 10

Distribution Parameters:

● Random

○ Stratified Random

○ Equalized Random

☐ Use Minimum points

Minimum Points: 10

Enter 10 here

The Distribution Parameters should be Random

Click here to start the process

2. In the Add Random Points dialog, enter a **10** in the **Number of Points** number field and press Return on your keyboard.

   In this example, you generate ten random points. However, to perform a proper accuracy assessment, you should generate 250 or more random points.

3. Confirm that the **Search Count** is set to **1024**.

   This means that a maximum of 1024 points are analyzed to see if they meet the defined requirements in the Add Random Points dialog. If you are generating a large number of points and they are not collected before 1024 pixels are analyzed, then you have the option to continue searching for more random points.

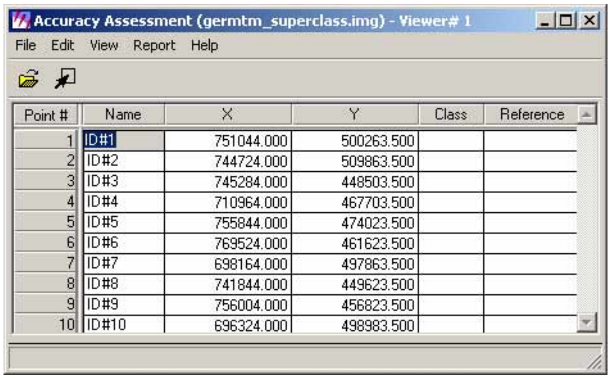
   NOTE: If you are having problems generating a large number of points, you should increase the **Search Count** *to a larger number.*

   The **Distribution Parameters** should be set to **Random**.

4. Click **OK** to generate the random points.

   The Add Random Points dialog closes and the Job Status dialog opens.

   This dialog automatically closes when the process is completed. A list of the points is shown in the Accuracy Assessment CellArray.



Point #	Name	X	Y	Class	Reference
1	ID#1	751044.000	500263.500		
2	ID#2	744724.000	509863.500		
3	ID#3	745284.000	448503.500		
4	ID#4	710964.000	467703.500		
5	ID#5	755844.000	474023.500		
6	ID#6	769524.000	461623.500		
7	ID#7	698164.000	497863.500		
8	ID#8	741844.000	449623.500		
9	ID#9	756004.000	456823.500		
10	ID#10	696324.000	498983.500		

5. In the Accuracy Assessment dialog, select **View | Show All**.

   All of the random points display in the **germtm.img** file in the Viewer. These points are white.

6. Analyze and evaluate the location of the reference points in the Viewer to determine their class value. In the Accuracy Assessment CellArray **Reference** column, enter your best guess of a reference relating to the perceived class value for the pixel below each reference point.

   As you enter a value for a reference point, the color of the point in the Viewer changes to yellow.

   If you were performing a proper accuracy assessment, you would be using ground truth data, previously tested maps, aerial photos, or other data.

7. In the Accuracy Assessment dialog, select **Edit | Show Class Values**.

The class values for the reference points appear in the **Class** column of the CellArray.

8. In the Accuracy Assessment dialog, select **Report | Options**. The **Error Matrix**, **Accuracy Totals**, and **Kappa Statistics** checkboxes should be turned on.

   The accuracy assessment report includes all of this information.

   See "Chapter 6: Classification" *of the* ERDAS Field Guide *for information on the error matrix, accuracy totals, and Kappa statistics.*

9. In the Accuracy Assessment dialog, select **Report | Accuracy Report**.

   The accuracy assessment report displays in the IMAGINE Text Editor.

10. In the Accuracy Assessment dialog, select **Report | Cell Report**.

    The accuracy assessment report displays in a second ERDAS IMAGINE Text Editor. The report lists the options and windows used in selecting the random points.

11. If you like, you can save the cell report and accuracy assessment reports to text files.

12. Select **File | Close** from the menu bars of both ERDAS IMAGINE Text Editors.

13. In the Accuracy Assessment dialog, select **File | Save Table** to save the data in the CellArray.

    The data are saved in the classified image file (**germtm_superclass.img**).

14. Select **File | Close** from the Accuracy Assessment dialog menu bar.

15. If you are satisfied with the accuracy of the classification, select **File | Close** from the Viewer menu bar.

    If you are not satisfied with the accuracy of the classification, you can further analyze the signatures and classes using methods discussed in this tour guide. You can also use the thematic raster layer in various ERDAS IMAGINE utilities, including the Image Interpreter, Raster Editor, and Spatial Modeler to modify the file.

Using the Grouping Tool

This section shows you how to use the Class Grouping Tool to assign the classes associated with an Unsupervised Classification and group them into their appropriate target classes. This tour is intended to demonstrate several methods for collecting classes, not to provide a comprehensive guide to grouping an entire Landsat image.

Setting Up a Class Grouping Project

In this example, we will take a Landsat image that has been classified into 235 classes using the ISODATA and the Maximum Likelihood classifications. These 235 classes will be grouped into a more manageable number of Land Use categories.

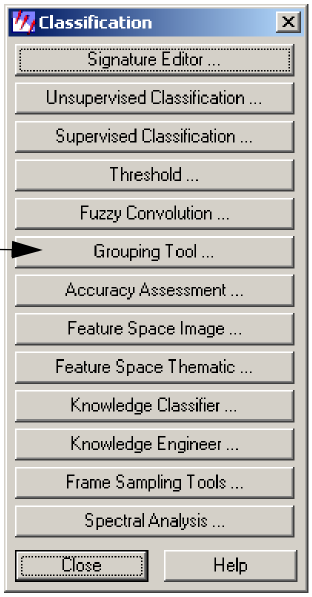
Starting the Class Grouping Tool

ERDAS IMAGINE should be running.

Copy the file **loudoun_maxclass.img** from the IMAGINE_HOME/examples directory into a directory in which you have write permission.

1. Click on the Classifier icon on the ERDAS IMAGINE icon panel.

   The **Classification** menu displays.


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Grouping Tool ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Frame Sampling Tools ...
Spectral Analysis ...
Close
Help

Click here to start the Grouping Tool

2. Select **Grouping Tool** from the **Classification** menu to start the Signature Editor.

   The **Select image to group** dialog opens.

3. Browse to the directory into which you copied the file **loudoun_maxclass.img**. Select it from the list of files and click **OK**.

   The **Class Grouping Tool** and a **Viewer** displaying the selected image file open.

4. To view the entire image, right-click in the **Viewer** and select **Fit Image to Window** from the Quick View menu.

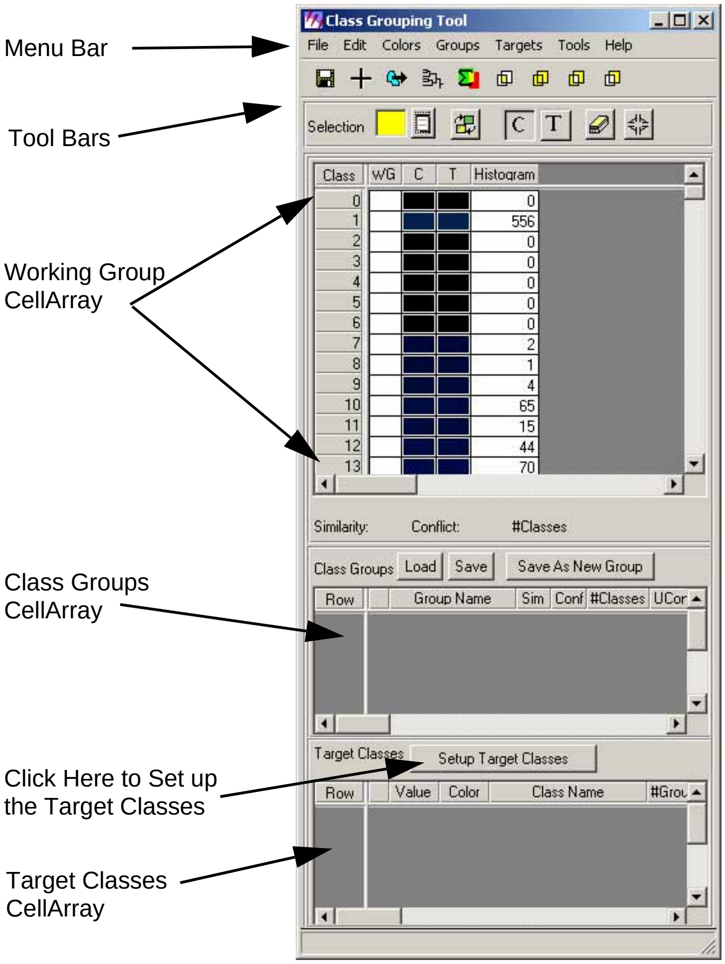
Class Grouping Tool Terminology

Classes are individual clusters of pixels with similar spectral characteristics. These clusters are the result of the unsupervised classification.

Target Classes are the final landuse or landcover categories for which you are interpreting.

Class Groups are the saved groups of classes that represent a single target class.

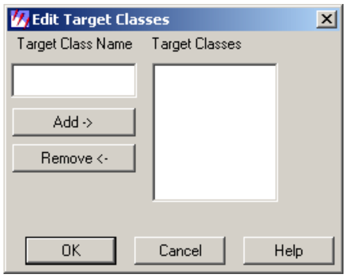
Working Groups are composed of the currently selected classes which are highlighted in the **Viewer**.


Menu Bar
Tool Bars
Working Group CellArray
Class Groups CellArray
Click Here to Set up the Target Classes
Target Classes CellArray
Class Grouping Tool
File Edit Colors Groups Targets Tools Help
Selection
C
T
Similarity: Conflict: #Classes
Class Groups Load Save Save As New Group
Row Group Name Sim Conf #Classes UCor
Target Classes Setup Target Classes
Row Value Color Class Name #Grou

Class	WG	C	T	Histogram
0		■	■	0
1		■	■	556
2		■	■	0
3		■	■	0
4		■	■	0
5		■	■	0
6		■	■	0
7		■	■	2
8		■	■	1
9		■	■	4
10		■	■	65
11		■	■	15
12		■	■	44
13		■	■	70

Set Up the Target Classes

1. Click the Setup Target Classes button above the Target Classes CellArray. the Edit Target Classes dialog opens.


Edit Target Classes
Target Class Name
Target Classes
Add ->
Remove <-
OK
Cancel
Help

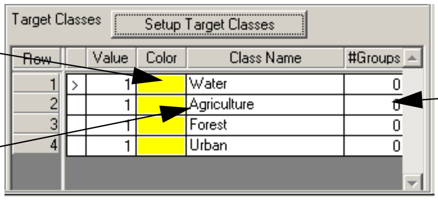
2. Place the Cursor in the Target Class Name field and type **Water**. Click the **Add ->** button.

   Water now appears in the list of Target Classes. Add Agriculture, Forest, and Urban classes. You may add as many additional classes as you wish.

3. Once you have finished adding Target Classes, click the **OK** button on the **Edit Target Classes** dialog.

NOTE: You may return to this dialog and add more Target Classes at any point during the Grouping process.

The Target Classes you have added appear in the Target Classes CellArray.



Row		Value	Color	Class Name	#Groups
1	>	1	🟨	Water	0
2		1	🟨	Agriculture	0
3		1	🟨	Forest	0
4		1	🟨	Urban	0

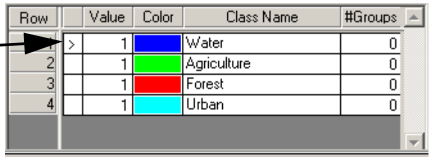
Click here to change the Target Color

Target Class Names

Displays the number of Class Groups in the Target Class

Now that the Target Classes are set up, you will assign Target colors.

4. Click in the Color Block next to the **Water** Target Class. Select Blue from the Color popup menu. Continue assigning colors to the Target Classes until colors have been assigned to each of them.



Row		Value	Color	Class Name	#Groups
	>	1	🟦	Water	0
2		1	🟩	Agriculture	0
3		1	🟥	Forest	0
4		1	🟦	Urban	0

The carat (>) indicates the currently selected Target Class

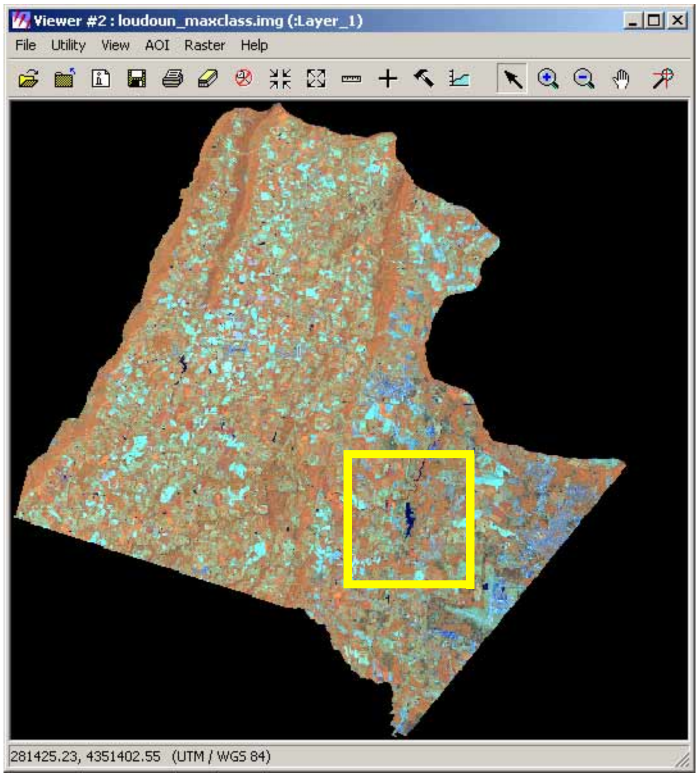
Collecting Class Groups

The main goal of a Class Grouping project is to gather classes of pixels which have common traits into the same Target Classes. To do this, you must select the classes and them save them to Class Groups. Class Groups are, as the name suggests, groups of classes that share similar traits; usually these are classes that are in the same land use category. The Class Groups are themselves members of the Target Classes into which the image is being stratified.

There are numerous ways to collect Class Groups. This tour will demonstrate how to use the Cursor Point Mode, the AOI Tool, and the Ancillary Data Tool to collect Class Groups.

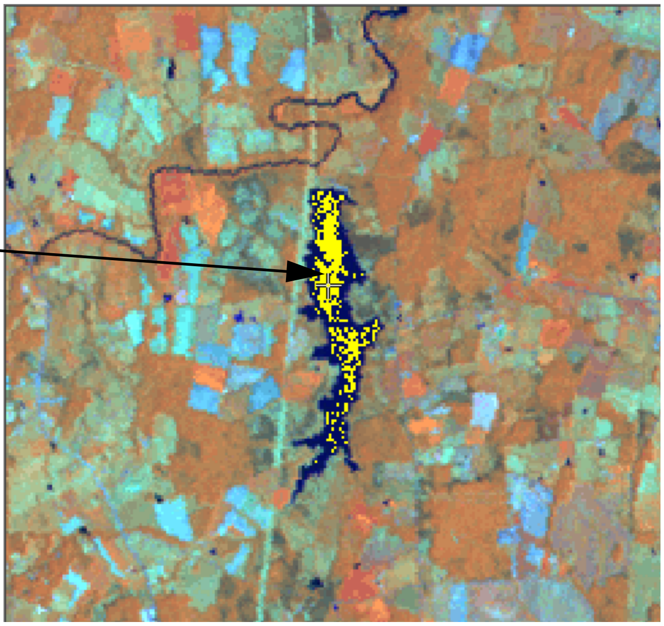
Using the Cursor Point Mode

1. In the Viewer, right-click and select **Inquire Cursor** from the Quick View menu. The Inquire Cursor dialog displays.
2. In the X: field, enter **280135.655592**. Enter **4321633.145953** into the Y: field. Press Enter.
3. Zoom in on the lake identified by the Inquire Cursor.


Viewer #2 : loudoun_maxclass.img (:Layer_1)
File
Utility
View
AOI
Raster
Help
281425.23, 4351402.55 (UTM / WGS 84)

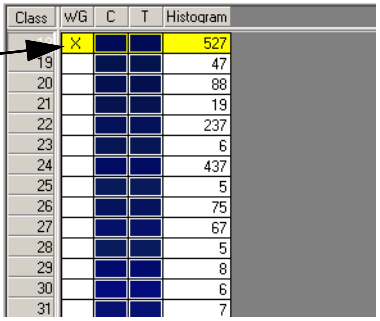
4. Click Close on the Inquire Cursor dialog.
5. Select the Cursor Point Mode icon + on the **Class Grouping Toolbar**. The cursor will change to a crosshair when it is placed in the Viewer.

6. In the Viewer, place the crosshair cursor over the lake and click.The lake, and all pixels which belong to the same classification as the pixel you selected, are highlighted in the Viewer.



All pixels in the same class are highlighted

The selected class is also highlighted in the Working Classes CellArray

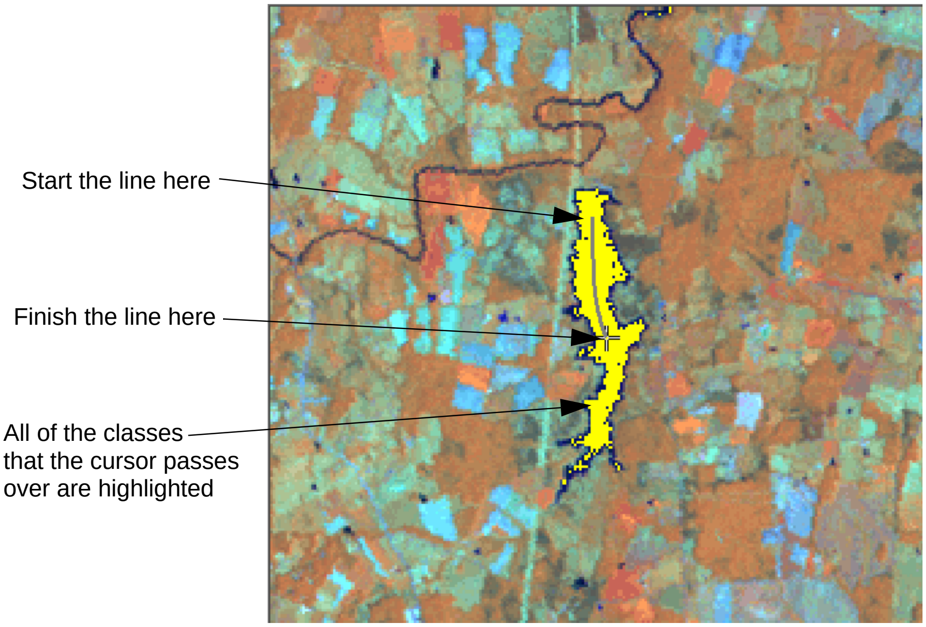


The Row is highlighted, and the WG (Working Group) column has an X indicating that this class is a member of the current Working Group.

Class	WG	C	T	Histogram
18	X	■	■	527
19		■	■	47
20		■	■	88
21		■	■	19
22		■	■	237
23		■	■	6
24		■	■	437
25		■	■	5
26		■	■	75
27		■	■	67
28		■	■	5
29		■	■	8
30		■	■	6
31		■	■	7

7. Click on the **X** in the WG column to clear the currently highlighted class from both the CellArray and the Viewer.

8. Now place the crosshair cursor over the lake. Click and drag the cursor in a short line over the lake. All of the classes that the cursor passes over will be selected in the Working Groups CellArray.



Start the line here

Finish the line here

All of the classes that the cursor passes over are highlighted

This provides a much better selection, but there is still some speckling in the selection.

9. Hold down the **Shift** key on the keyboard, and then click on one of the unselected pixels. Note this adds all of the pixels to the currently selected classes in the Working Group.

10. Now hold down the **Ctrl** key on the keyboard and click on one of the highlighted pixels. All of the pixels that belong to the same class as this pixel are removed from the selection.

    NOTE: The Shift and Ctrl keys may also be used to select and deselect classes directly in the Working Classes CellArray.

Filling in the Holes and Removing the Speckle

The initial step in any collection method will leave either holes--unselected classes that are “islands” within the class--or speckles--selected classes that are “islands” outside of the majority of the selected classes. To increase the accuracy of your Class Groups, you will need to fill the holes and remove the speckle.

11. Continue to collect the water classes of this lake using the **Shift** and **Ctrl** keys. Use the Toggle Highlight button to turn off the highlighting and see the actual pixels you have selected.

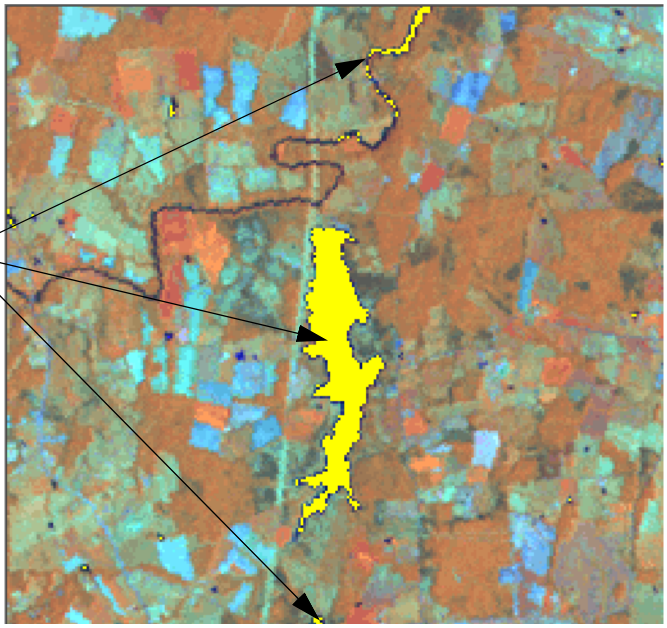
Include the class if:

- adding the class fills the holes in the existing selection,
- adding the class supplements the edges of the existing selection,
- removing the class opens significant holes in the selection, or
- adding the class reduces the overall complexity of the selection.

Exclude the class if

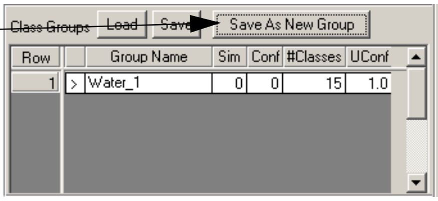
- adding the class creates speckles in places where there were none before,
- removing the class removes speckles in the overall image,
- removing the class reduces the overall complexity of the selection

Your selections will look similar to this:



Note that all of the pixels that belong to these classes are selected

12. Save the Working Group as a Class Group by clicking the **Save As New Group** button.



Row		Group Name	Sim	Conf	#Classes	UConf
1	>	Water_1	0	0	15	1.0

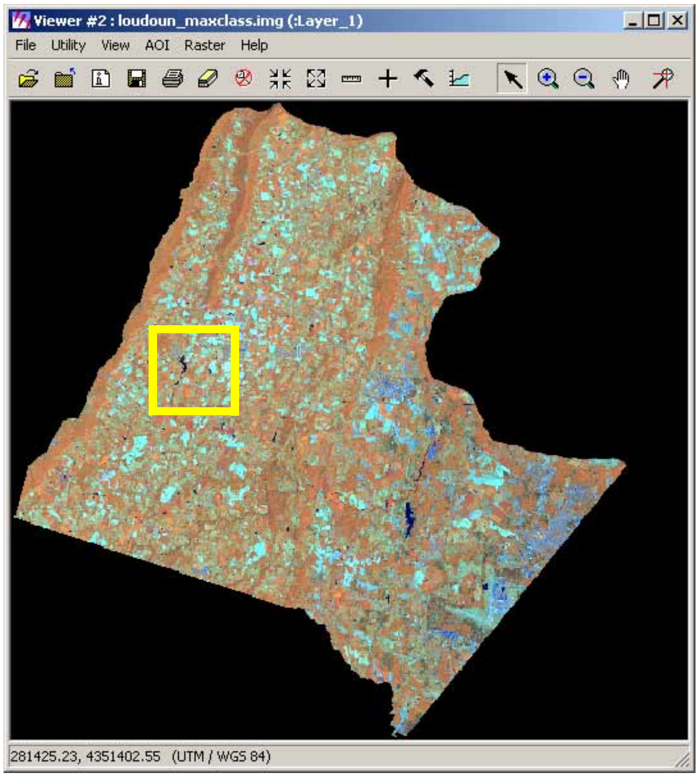
Click here to save
the Working Group
as a new Class Group
The new Class Group is

Using the AOI Tools

1. In the Viewer, right-click and select **Inquire Cursor** from the Quick View menu. The **Inquire Cursor** dialog displays.

2. In the Inquire Cursor dialog, enter **261278.630592** in the X: field and **4334243.327665** in the Y: field.

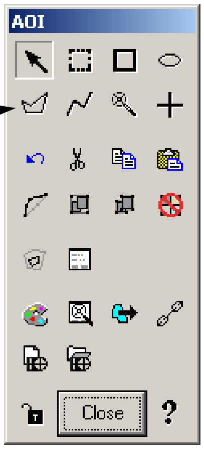
3. Use the re-centering zoom tool to zoom in on the lakes identified by the Inquire Cursor.,


Viewer #2 : loudoun_maxclass.img (:Layer_1)
File
Utility
View
AOI
Raster
Help
281425.23, 4351402.55 (UTM / WGS 84)

4. Click **Close** on the **Inquire Cursor** dialog to dismiss it.

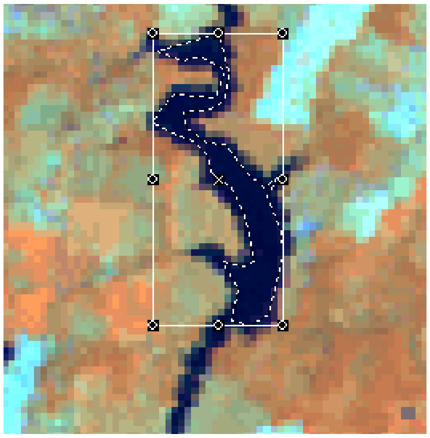
5. If the Class Group from the previous section is still highlighted in the Viewer, use the Clear Working Group contents button to clear the selections.

6. Select **AOI | Tools** from the **Viewer** menu bar. The **AOI Tools** palette is displayed.



Select the Polygon Tool

7. Digitize a polygon the encompasses the majority of the open water pixels in the largest lake.‘



8. In the Class Grouping Tool toolbar, click the Use Current AOI to Select Classes button.

   All of the classes that are contained within the currently selected AOI are highlighted in the Working Groups CellArray.

9. Using the techniques outlined in "Using the Cursor Point Mode" on page 504, fill in the holes in the selections for these lakes.

10. In the Class Groups CellArray, make sure that the carat > is in the row for the **Water_1** class and click the **Union** button.

    This adds the **Water_1** Class Group to the classes that are currently selected in the Working Classes CellArray.

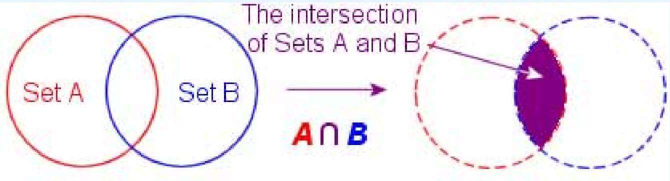
11. In the Class Groups CellArray, click in the **Water_1** cell. This group represents the open water land use category, so change the Group Name to **Open**.

    NOTE: The Target Class Name is already a stored part of the Class Group name, so there is no need to repeat it in the Class Group name.

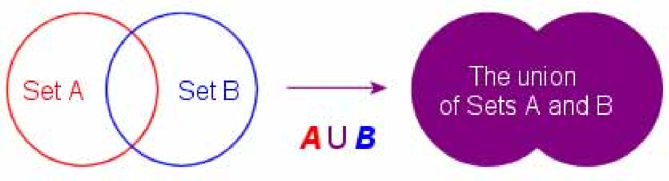
Definitions of Boolean Operators

The Class Grouping Tools provides four boolean operators that allow you to refine the selections in your Class Groups.

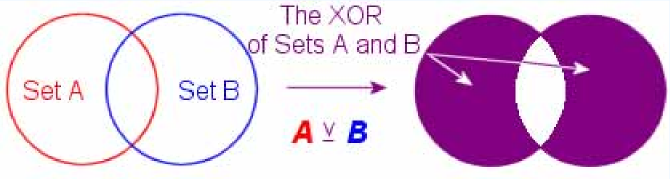
Intersection of Sets: The intersection of two sets is the set of elements that is common to both sets.


The intersection
of Sets A and B
Set A
Set B
A ∩ B

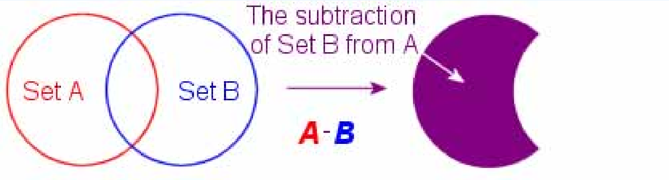
Union of Sets: The union of two sets is the set obtained by combining the members of each set.


Set A
Set B
A ∪ B
The union
of Sets A and B

Exclusive-Or (XOR) of Sets: The Exclusive-Or of two sets is the set of elements belonging to one but not both of the given sets.


The XOR
of Sets A and B
Set A
Set B
A ⊻ B

Subtraction of Sets: The subtraction of set B from set A yields a set that contains all data from A that is not contained in B.e saved groups of classes that represent a single target class.


The subtraction
of Set B from A
Set A
Set B
A - B

Using the Ancillary Data Tool

It would take a very long time to collect all of the classes in a large image using only the simple tools outlined above. To save time, you should quickly group all of the classes into Class Groups and then refine these initial groupings to more accurately define the study area.

The **Ancillary Data Tool** provides a means of performing this quick initial grouping. By using previously collected data, such as ground truth data or a previous classification of the same area, you can quickly group your image, and then concentrate on evaluating and correcting the groups.

The thematic file used as the Ancillary Data file need not cover the entire area, but it must at least overlap with the area being grouped.

Setting Up the Ancillary Data Classes

1. If the Class Group from the previous section is still highlighted in the Viewer, use the Clear Working Group contents button to clear the selections.

2. In the Class Grouping Tool, click the Start Ancillary Data Tool button.

   Two dialogs are displayed, the **Ancillary Data Tool** dialog and the **Ancillary Class Assignments** dialog.

3. In the **Ancillary Class Assignments** dialog, select **File | Set Ancillary Data Layer**. The File Chooser opens.

4. Select **loudoun_lc.img** from the **IMAGINE_HOME/examples** directory. Click OK. A Performing Summary progress meter appears. When the summary is complete, click **OK** to dismiss the progress meter.

   The summary process will do three things:

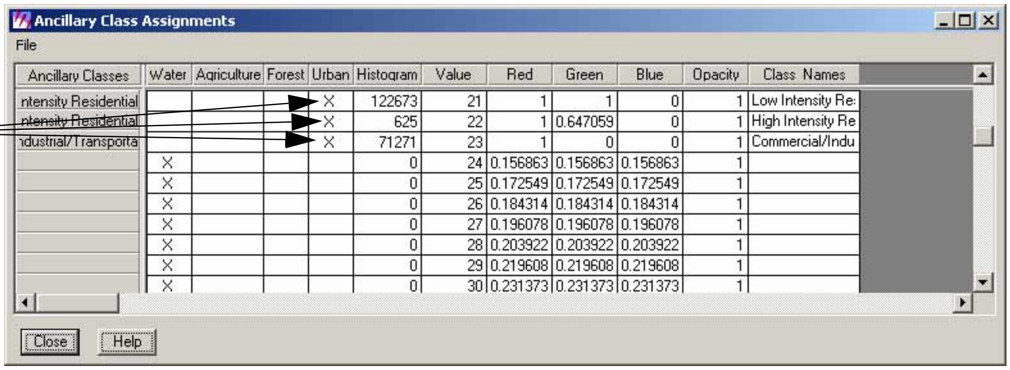
   - populate the **Ancillary Class Assignments** CellArray with information from the ancillary data file,
   - provide summary values relating the ancillary data file to the file being grouped in the **Ancillary Data Tool** CellArray,
   - add three new column (Diversity, Majority, and Target %) to the Working Groups CellArray in the **Class Grouping Tool**.

For a more detailed explanation of each of these dialogs and their contents, please see the ERDAS IMAGINE On-Line Help.

In the **Ancillary Class Assignments** CellArray, the rows represent the classes from the ancillary data file (loudoun_lc.img) and the columns represent the information from the being grouped (loudoun_maxclass.img).

5. In the **Ancillary Class Assignments** CellArray, scroll down until you find the Low Intensity Residential class. Click in the Urban column of the CellArray to assign this class to the Urban Target Class. The **X** will move from the Water column (or the first column in the CellArray) to the Urban column.

6. Repeat this step for the High Intensity Residential and Commercial/Industrial/Transportation classes.



Ancillary Classes	Water	Agriculture	Forest	Urban	Histogram	Value	Red	Green	Blue	Opacity	Class Names
ntensity Residential				X	122673	21	1	1	0	1	Low Intensity Re:
ntensity Residential				X	625	22	1	0.647059	0	1	High Intensity Re
ıdustrial/Transporta				X	71271	23	1	0	0	1	Commercial/Indu
	X				0	24	0.156863	0.156863	0.156863	1	
	X				0	25	0.172549	0.172549	0.172549	1	
	X				0	26	0.184314	0.184314	0.184314	1	
	X				0	27	0.196078	0.196078	0.196078	1	
	X				0	28	0.203922	0.203922	0.203922	1	
	X				0	29	0.219608	0.219608	0.219608	1	
	X				0	30	0.231373	0.231373	0.231373	1	

Click here to relate
the Ancillary Data classes
to the Target Classes
in the image being grouped

7. Continue arranging the **X**s in the Ancillary Class Assignments so that they properly relate the classes from the ancillary data file to the Target Classes. If the ancillary data classes do not have labels, leave them in the Water column.

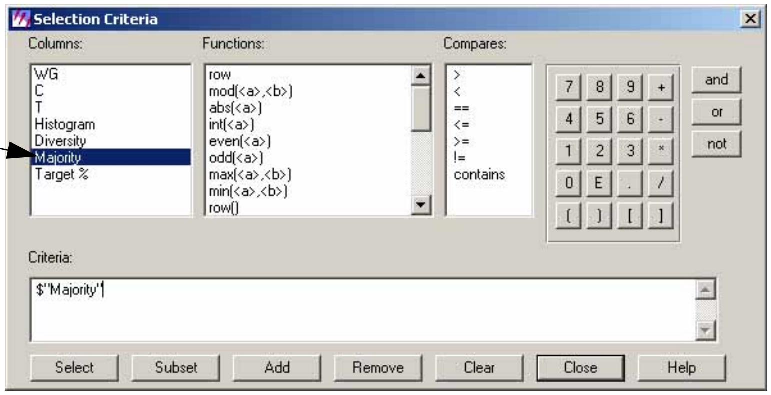
Collecting Groups Using the Majority Approach

In most cases, this approach would be the first step you took in the grouping process. As a first step, this process would result in a completely grouped image that had no Similarities and no Conflicts between Target Classes.

We have already begun collecting Class Groups, and this will cause some conflicts between Target Classes.

Once you have assigned the ancillary data classes to the Target Classes, you may minimize the **Ancillary Data Tool** and the **Ancillary Class Assignments** dialogs.

1. In the Working Classes CellArray on the **Class Grouping Tool**, right-click in the row labels. Select **Criteria...** from the Row Selection menu that appears. The **Selection Criteria** dialog is displayed.

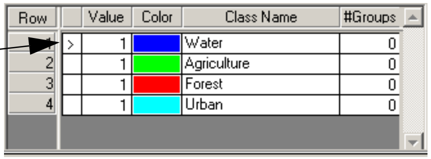

Selection Criteria
Columns:
WG
C
T
Histogram
Diversity
Majority
Target %
Functions:
row
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
max(<a>,<b>)
min(<a>,<b>)
row()
Compares:
>
<
==
<=
>=
!=
contains
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
and
or
not
Select
Subset
Add
Remove
Clear
Close
Help

Criteria: $"Majority"

Click Majority to specify the selection criteria

2. In the **Columns:** area, click **Majority** to set the selection criteria.

3. In the **Class Grouping Tool**, select the Water Target Class by placing the carat > in the Water Row

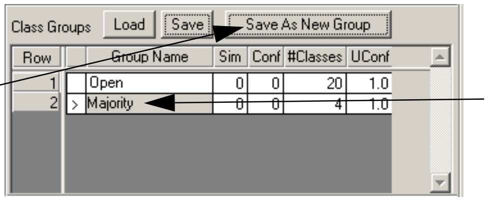


Row		Value	Color	Class Name	#Groups
	>	1	🟦	Water	0
2		1	🟩	Agriculture	0
3		1	🟥	Forest	0
4		1	▨	Urban	0

Make sure that the carat > is in this row

4. In the **Selection Criteria** dialog, click the **Select** button. All of the classes that best represent the selected Target Class are highlighted.

5. In the Class Groups area of the **Class Grouping Tool**, click the **Save As New Group** button. The Selected Classes are added as a new Class Group.


Class Groups
Load
Save
Save As New Group

Row		Group Name	Sim	Conf	#Classes	UConf
1		Open	0	0	20	1.0
2	>	Majority	0	0	4	1.0

Click here to save the classes as a new group

Change the name of the new group to Majority

6. Change the name of the new group to **Majority**. This will help keep track of how the Class Group was collected.

7. Repeat steps 3. through 6. for each of the Target Classes, moving the carat in the Target Classes CellArray to the next Target Class each time.

8. Save the Grouping Process by selecting **File | Save Image...** from the **Class Grouping Tool** menu bar.

   This provides a broad grouping of all the classes in the image, and each Class Group must be closely examined to determine the accuracy of the Majority grouping.

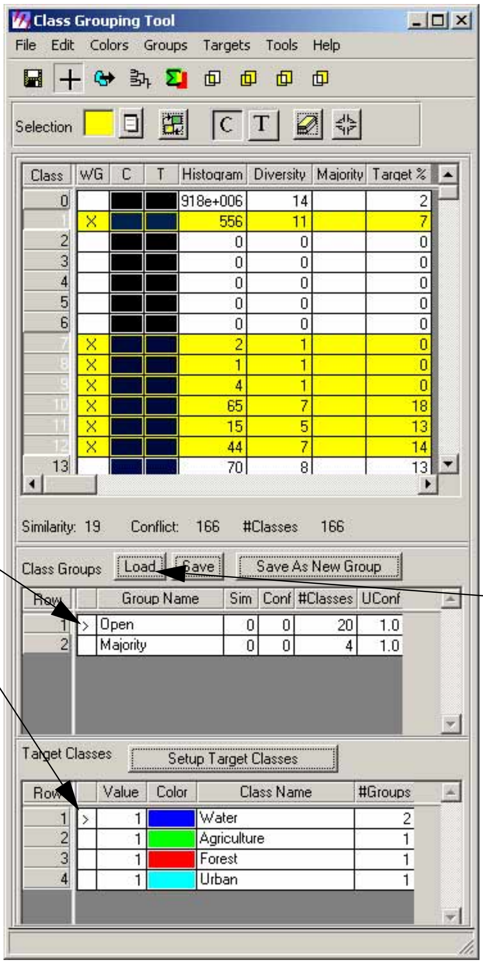
   Next we will discuss how to find the grouping conflicts and some strategies for resolving them.

Identifying and Resolving Similarities and Conflicts

The Class Grouping Tool allows there to be any number of Class Groups representing each Target Class, and there is no limit on whether these groups overlap or conflict with each other. It is frequently the case that a single class may properly belong with more than one target class. These classes are termed conflicted classes, and they generally are a source of speckle in the resulting final classification.

Both Similarity and Conflict are measures of shared classes. Similar classes are shared by other groups within the same Target Class, while conflicted classes shared by groups under a different Target Class.

1. If the classes from a previous section are still highlighted in the Viewer, use the Clear Working Group Contents button to clear them.
2. In the **Class Grouping Tool**, select the Water Target Class by placing the carat > in the Water row. Select the Open Class Group by placing the carat in the Open row.


Class Grouping Tool
File
Edit
Colors
Groups
Targets
Tools
Help
Selection
C
T
Class Groups
Load
Save
Save As New Group
Target Classes
Setup Target Classes

Class	WG	C	T	Histogram	Diversity	Majority	Target %
0				918e+006	14		2
1	X			556	11		7
2				0	0		0
3				0	0		0
4				0	0		0
5				0	0		0
6				0	0		0
7	X			2	1		0
8	X			1	1		0
9	X			4	1		0
10	X			65	7		18
11	X			15	5		13
12	X			44	7		14
13				70	8		13

Similarity: 19 Conflict: 166 #Classes 166

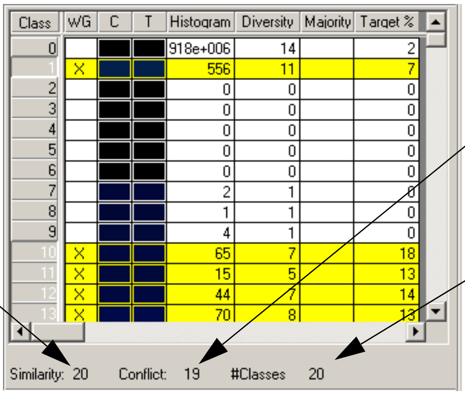
Row		Group Name	Sim	Conf	#Classes	UConf
1	>	Open	0	0	20	1.0
2		Majority	0	0	4	1.0

Row		Value	Color	Class Name	#Groups
1	>	1	🟦	Water	2
2		1	🟩	Agriculture	1
3		1	🟥	Forest	1
4		1	🟦	Urban	1

Place the carats > here...

...and then click Load to highlight the classes in the Working Groups CellArray

Notice the Similarity and Conflict numbers that are displayed just under the Working Groups CellArray:



Class	WG	C	T	Histogram	Diversity	Majority	Target %
0				918e+006	14		2
1	X			556	11		7
2				0	0		0
3				0	0		0
4				0	0		0
5				0	0		0
6				0	0		0
7				2	1		0
8				1	1		0
9				4	1		0
10	X			65	7		18
11	X			15	5		13
12	X			44	7		14
13	X			70	8		13

Similarity: 20 Conflict: 19 #Classes 20

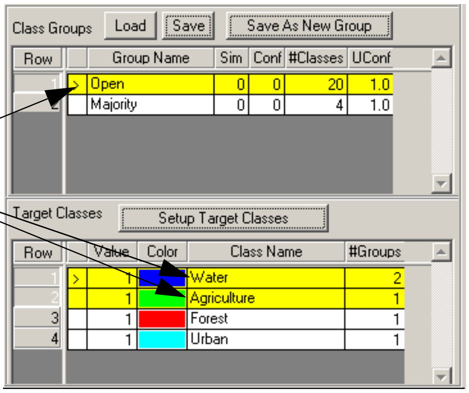
Number of similarities between the Working Group and the selected Class Group

Number of conflicts between the Working Group and the selected Class Group

Number of selected classes in the Working Group CellArray

The Similarity statistics are calculated between the Working Groups and the selected Class Group. These are the same, so we would expect there to be the same number of Similarities and Classes Selected, and this is the case.

3. Click the **Toggle Similarity/Conflict** button to highlight the other Target Classes that have classes in common with the Working Group (which is exactly the same as the Open Class Group).



Row		Group Name	Sim	Conf	#Classes	UConf
1	>	Open	0	0	20	1.0
2		Majority	0	0	4	1.0

Row		Value	Color	Class Name	#Groups
1	>	1	🟦	Water	2
2		1	🟩	Agriculture	1
3		1	🟥	Forest	1
4		1	■	Urban	1

The Target Classes and Class Groups that share classes are highlighted

4. Select the Agriculture Target Class by placing the caret in the Agriculture row.

   NOTE: The contents of the Working Group CellArray do not change when you change the selected Target Class, but the contents of the Similarity and Conflict statistics have reversed.

5. To identify the classes that these two Target Classes have in common, click the intersection boolean button.

   For more information on Boolean operators, see "Definitions of Boolean Operators" on page 511.

   This loads the intersection of the classes included in the Working Group (Water, Open) and the selected Class Group (Agriculture, Majority).

6. In the **Viewer**, zoom in on the classes that are currently selected. These classes are located in the Lakes we collected with the AOI Tool.

7. Use the Toggle Highlighting button to view the pixels in question. These pixels belong in the Water Target Class and not in the Agriculture Target Class.

8. To remove these classes from the Agriculture Target Class, make sure the caret is still in the Agriculture Target Class and the Majority Class Group.

9. Click the **Exclusive-Or** boolean button . This loads all of the classes in the selected Class Group (Agriculture, Majority) without any of the classes that were previously highlighted in the CellArray (the conflicted classes).

10. In the Class Groups area, click the **Save** button to save the Agriculture, Majority Class Group without the conflicted classes.

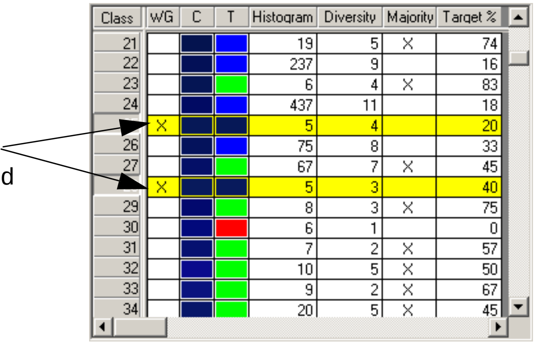
11. In the Class Grouping Tool menu bar, click the Save button to save the grouping process to the image file.

Coloring the Thematic Table

Sometimes it is helpful to judge your progress by seeing the entire picture. The Class Grouping Tool provides a mechanism for you to see how the grouping process is progressing.

1. In the Class Grouping Tool menu bar, select **Colors | Set to Target Colors**.

   The colors in the **T** column in the Working groups CellArray will change to reflect the colors of the associated Target Classes.



Classes that are conflicted or are not included in any Target Class are highlighted

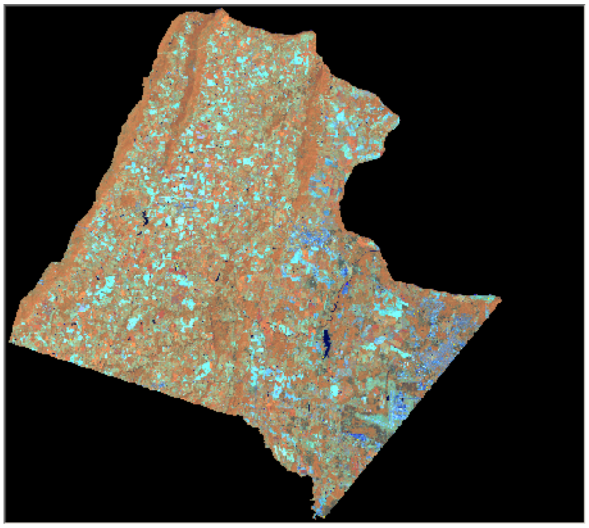
Class	WG	C	T	Histogram	Diversity	Majority	Target %
21		🟦	🟦	19	5	X	74
22		🟦	🟦	237	9		16
23		🟦	🟩	6	4	X	83
24		🟦	🟦	437	11		18
	X	🟦	🟦	5	4		20
26		🟦	🟦	75	8		33
27		🟦	🟩	67	7	X	45
	X	🟦	🟦	5	3		40
29		🟦	🟩	8	3	X	75
30		🟦	🟥	6	1		0
31		🟦	🟩	7	2	X	57
32		🟦	🟩	10	5	X	50
33		🟦	🟩	9	2	X	67
34		🟦	🟩	20	5	X	45

2. All of the classes that have not been grouped into Target Classes or are in conflict with other Target Classes will be highlighted.

3. Refine the groupings to include all of the highlighted classes.

4. To change back to the standard color table, click the C button. To view the Thematic color table, click the T button.

5. When you have finished the grouping process, click the T button to display the Thematic colors in the Viewer, then select **File | Save | Top Layer As...**. Save the image as **loudoun_strata.img**.



CHAPTER EIGHTEEN

Frame Sampling Tools

Introduction

Lets say that you needed to assess the amount a land that is covered by parking lots on a university campus. How would you go about accomplishing this? You could either go and start surveying parking lots or you could get aerial photography of the campus and start digitizing them. But what if you wanted to analyze the amount of land covered by forests in an entire county or the amount of arable land planted with grain in an entire state? The cost of collecting ground truth data from the entire county or of digitizing the entire state would be prohibitive to getting an accurate assessment.

The process of Frame Sampling provides an answer. Frame Sampling is a statistical methodology that enables the accurate survey of a Material of Interest (MOI) in the study area. As the name suggests, Frame Sampling uses a frame to define the study area and the analysis of representative samples from within that frame to estimate the proportion of the MOI in the frame. Although getting ground truth from an entire county or digitizing and entire state might not be feasible, it would certainly make sense to use ground truth and imagery interpretation to calculate the amount of the MOI in these representative samples.

Remote Sensing and Frame Sampling

The use of Frame Sampling and remote sensing can assist the surveyor in achieving the most accurate estimate for the least cost. Remote Sensing provides the user with a synoptic view of the entire Frame.

The classification methods described in "Perform Unsupervised Classification" on page 483 and the Class Grouping Tool described in "Using the Grouping Tool" on page 500 provide two methods of "stratification" or creating smaller homogenous units that represent the entire Frame. This stratification reduces the number of samples that allocated to provide an accurate result.

High resolution aerial photography can be used in the labeling of the ares containing the MOI in the representative samples, thereby limiting the amount of ground truth data that needs to be collected.

Frame Sampling Tools

The Frame Sampling Tools provide a framework guiding the Frame Sampling process, a means of managing the array of files generated by the process, links to the appropriate remote sensing tools, and the necessary computations for the Final Analysis of the MOI.

Frame Sampling Tools Tour

This Tour is intended to walk you through a landcover analysis Frame Sampling project.The frame for this project is defined by the political boundaries of Loudon County. The MOI for this project is forest cover.

The Frame Sampling Project Manager provides the ability to track and perform the necessary steps for preparing a file for Frame Sampling. For the purposes of this Tour, the following preparatory steps have already been performed for you:

- Obtain a large-scale synoptic image (or images) that covers the entire study area
- Orthorectify that synoptic image. OrthoRecitifaction is explained in "Orthorectification" on page 265
- Classify the orthorectified image using a classification technique such as the ISODATA classification described in "Perform Unsupervised Classification" on page 483
- Group the Classified image with the Class Grouping Tool. Tips an techniques for Grouping the Classified image are illustrated in "Using the Grouping Tool" on page 500

Setting Up the Sampling Project

This section shows you how to set up a Sampling Project and how the project manager allows you to manage and track the files used in the Frame Sampling process.

You perform the following operations in this section:

- Create a Sampling Project
- Assign the Project Files
- Recode the Grouped File
- Create a Sampling Grid
- Select the Samples for Interpretation
- Use the Dot Grid Tool to Label the Samples
- Compute the Final Analysis and Fraction Files

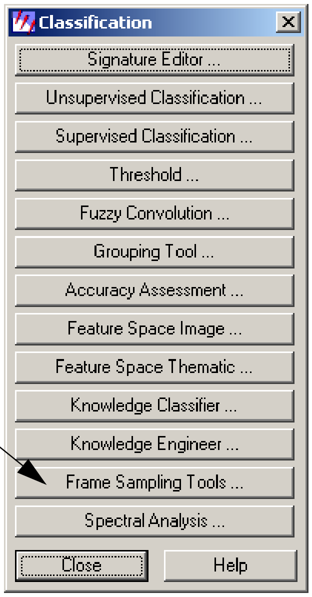
ERDAS IMAGINE must be running.

Create a New Sampling Project

The first step in the Frame Sampling process is to create the sampling project.

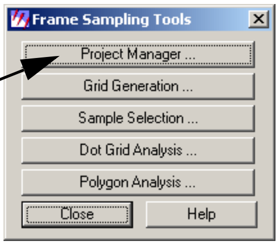
1. Click on the Classifier icon Classifier on the ERDAS IMAGINE icon panel.

The **Classification** menu displays.


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Grouping Tool ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Frame Sampling Tools ...
Spectral Analysis ...
Close
Help

Click here to open the Frame Sampling Tools menu

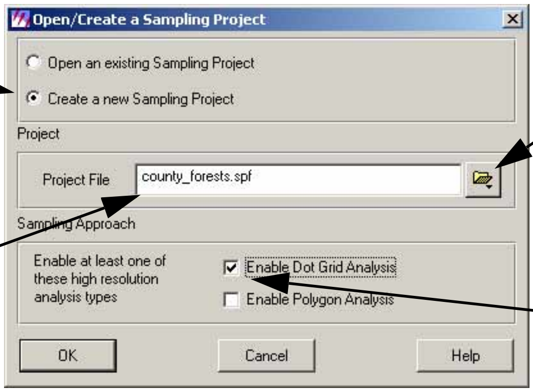
2. Click the Frame Sampling Tools button to open the **Frame Sampling Tools** menu.


Frame Sampling Tools
Project Manager ...
Grid Generation ...
Sample Selection ...
Dot Grid Analysis ...
Polygon Analysis ...
Close
Help

Click here to open the Project Manager

3. Click the **Project Manager** button on the **Frame Sampling Tools** menu.

   The **Open/Create a Sampling Project** dialog opens.


Open/Create a Sampling Project
Project
Sampling Approach
OK
Cancel
Help

○ Open an existing Sampling Project
● Create a new Sampling Project

Project File: county_forests.spf

Enable at least one of these high resolution analysis types
☑ Enable Dot Grid Analysis
☐ Enable Polygon Analysis

Click here to create a new Sampling Project

Enter the project name

Browse to a directory in which you have write permission

Click here to enable Dot Grid Analysis

4. Click the Create a new Sampling Project radio button.

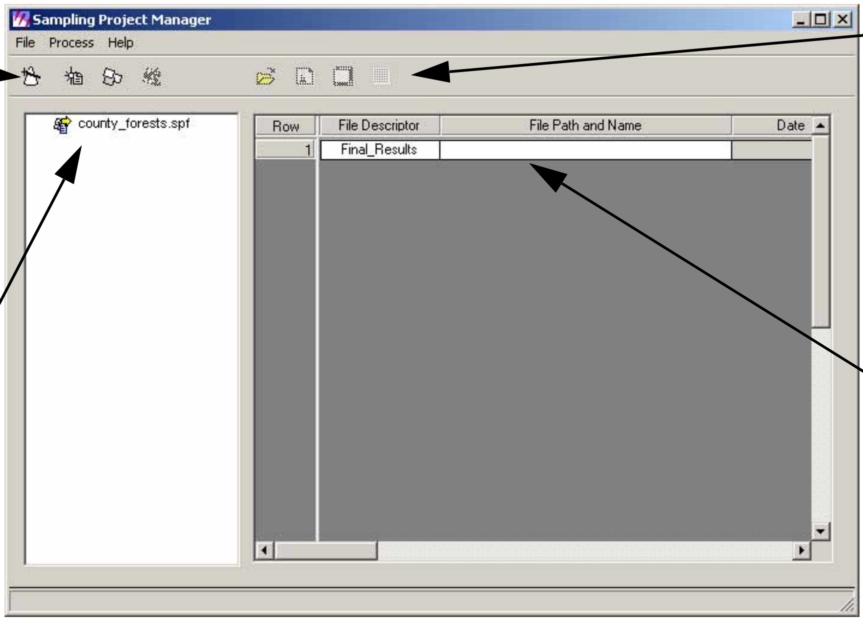
5. Browse to a directory in which you have write permission.

6. Enter **county_forests.spf** as the project name.

7. Select the **Enable Dot Grid Analysis** checkbox.

8. Click **OK**.

The Sampling Project Manager opens displaying the contents of your new project.


Sampling Project Manager
File
Process
Help
county_forests.spf
Row
File Descriptor
File Path and Name
Date
1
Final_Results

These buttons affect the selected node in the Tree View

These buttons affect the file currently selected in the Files CellArray

The Tree View is a hierarchical view of all files in the project

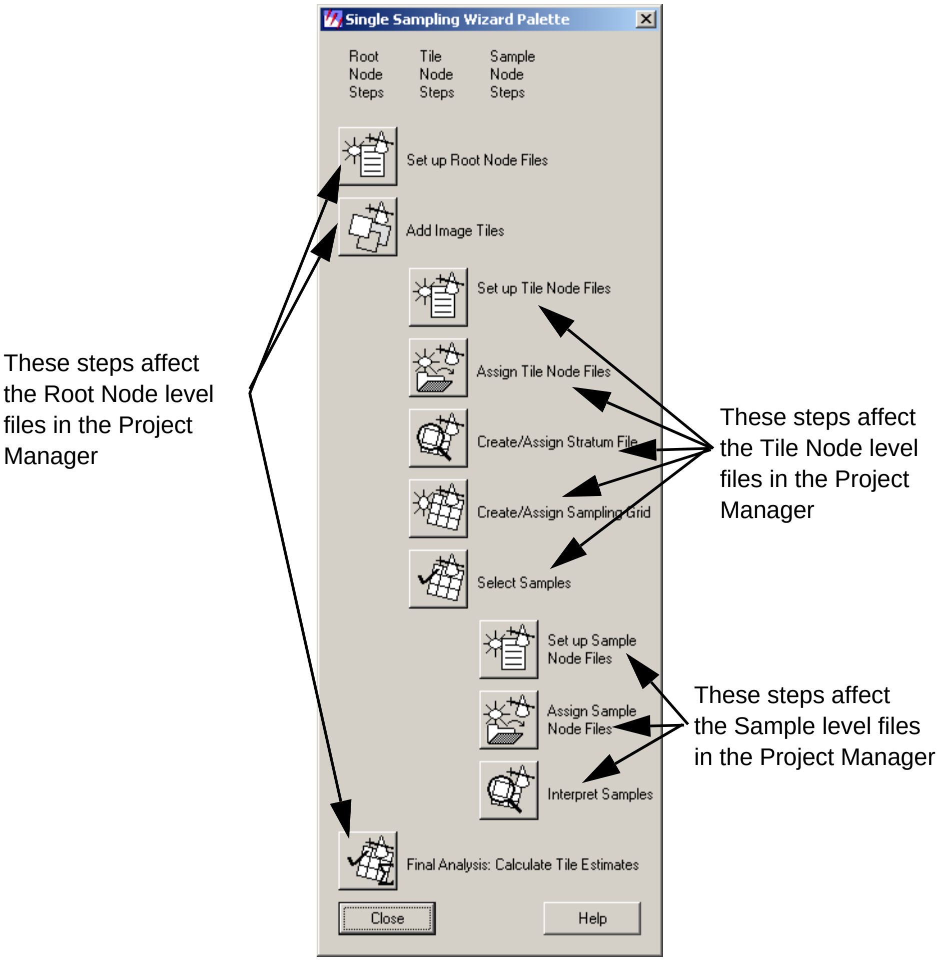
The Files CellArray shows all the files associated with the item selected in the Tree View

The Frame Sampling process is very long, and can take several days for large projects. You can save your progress on any project by selecting **File | Save** *from the Project Manager menu bar. You may now exit the project and return to it without losing any progress.*

Root Level Functions

The Single Sampling wizard is designed to walk you through the steps associated with the Frame Sampling process.

1. Click the Use Sampling Project Wizard button to open the **Single Sampling Wizard Palette**.


Single Sampling Wizard Palette
Root Node Steps
Tile Node Steps
Sample Node Steps
Set up Root Node Files
Add Image Tiles
Set up Tile Node Files
Assign Tile Node Files
Create/Assign Stratum File
Create/Assign Sampling Grid
Select Samples
Set up Sample Node Files
Assign Sample Node Files
Interpret Samples
Final Analysis: Calculate Tile Estimates
Close
Help

These steps affect the Root Node level files in the Project Manager

These steps affect the Tile Node level files in the Project Manager

These steps affect the Sample level files in the Project Manager

The general workflow in the Frame Sampling process will move from the top to the bottom of the Palette. Clicking on a button in the palette will jump you directly to that step in the Frame Sampling workflow wizard.

Single Sampling Project Nodes

Root Node Level steps affect the project as a whole. Root Node files will appear in the far left of the Tree View hierarchy.

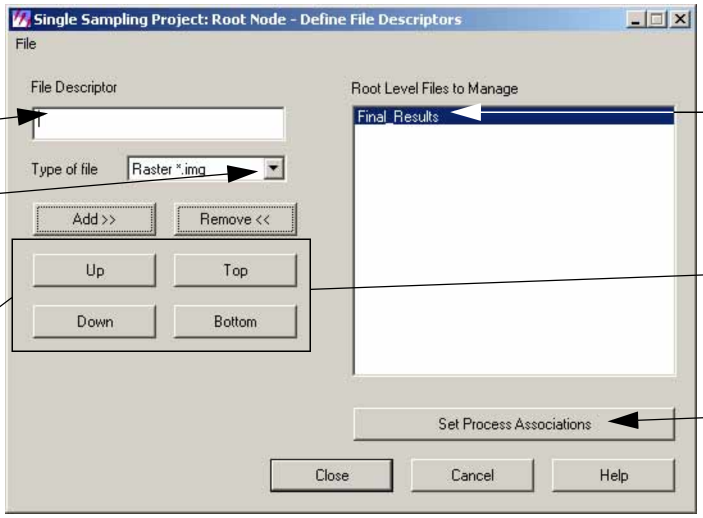
Tile Node Level steps are performed on the Tiles. Image Tiles are the large-scale synoptic images that cover the study area. Each of these Tiles will be stratified and then divided into representative samples. Tile Node files are dependent upon one of the Root Node files.

Sample Node Level steps are performed on the high-resolution representative samples of the Image Tiles. These Samples are interpreted for the MOI. Sample level files are dependent upon one of the Image Tiles.

2. Click the Set Up Root Node Files button on the **Single Sampling Wizard Palette** to open the first step of the **Single Sampling Wizard**.

   The Single Sampling Wizard opens. Each step in the Wizard has text that explains the current step and either allows you to set up the file assignments or click buttons to launch the appropriate tools.

3. Click the Setup Files button in the **Setup Root Node Files** step of the wizard. The Root Node - Define File Descriptors dialog is displayed.


Single Sampling Project: Root Node - Define File Descriptors

Type of file: Raster *.img

Root Level Files to Manage: Final_Results

To Add a New File:

Enter the Descriptor for the new file

Select the type of file to add

Click **Add >>**

Modify the location of the new file in the project hierarchy

To Edit an Existing File:

Select a descriptor from the list of root level files

Modify the location of the existing file in the project hierarchy

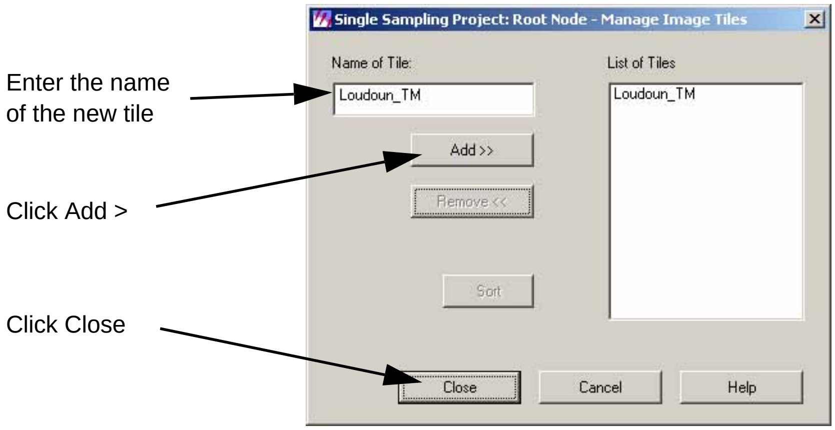
Click to set the relationship between the selected file and the other files in the project

This dialog allows you to manage the files in the Sampling Project. You can add and remove files from the process, as well as modify the relationship between a file and the process by clicking the **Set Process Associations** button.

4. Click **Close** without making any changes to the files. You are returned to the Single Sampling Wizard.

5. Click **Next >** on the Single Sampling Project Wizard. The **Add Image Tiles** step is displayed.

6. Click the Add Image Tile button on the Wizard. The **Manage Image Tiles** dialog is displayed.



Single Sampling Project: Root Node - Manage Image Tiles

Name of Tile: Loudoun_TM

List of Tiles: Loudoun_TM

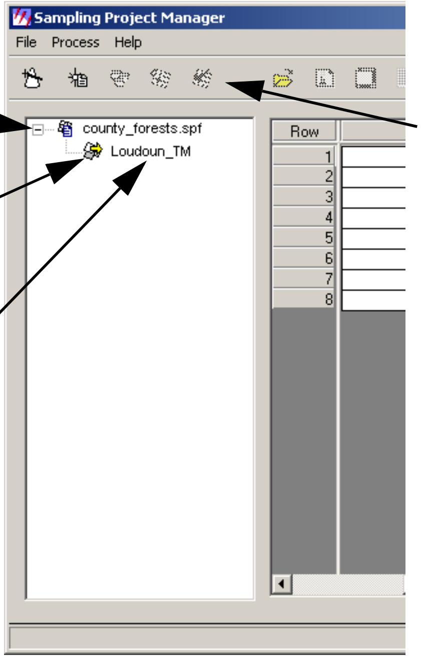
Enter the name
of the new tile
Click Add >
Click Close
Add >>
Remove <<
Sort
Close
Cancel
Help

This dialog allows you to add names for the tiles in the Sampling project.

The Image Tiles must cover the entire area frame. You may need to add more than one tile if the frame cannot be covered by a single tile.

7. In the Name of Tile field, type **Loudoun_TM**.
8. Click **Add >** to add the Tile name to the List of Tiles.
9. Click **Close** to exit the **Manage Image Tiles** dialog.

Note that the Tile Node has been added to the Tree View in the **Project Manager**.


Sampling Project Manager
File Process Help
county_forests.spf
Loudoun_TM
Row
1
2
3
4
5
6
7
8

Click here to expand the Root Node

The arrow indicates the node that is currently selected

Click on the new Tile name to display files associated with Tile-level processes

The buttons on the toolbar change to provide access to Tile-level functions

10. Click **Next** on the Wizard. The Set Up Tile Node level files step displays.

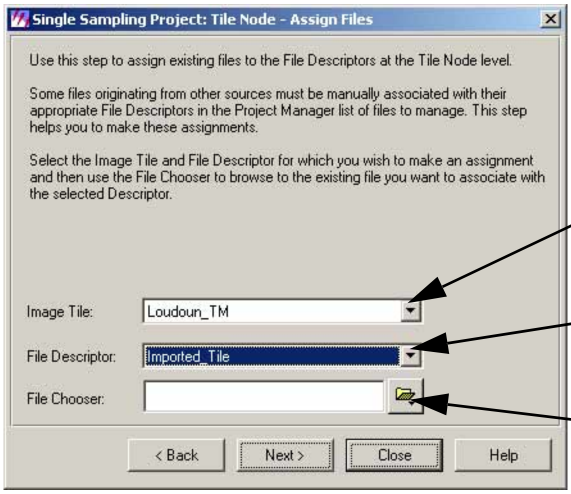
Tile Level Functions

The Tile level functions are processes that apply to the entire Image Tiles.

1. On the **Tile Node - Set Up Files** step, click **Next**.

The **Assign Tile Node Files** step displays in the Wizard

.


Single Sampling Project: Tile Node - Assign Files

Use this step to assign existing files to the File Descriptors at the Tile Node level.

Some files originating from other sources must be manually associated with their appropriate File Descriptors in the Project Manager list of files to manage. This step helps you to make these assignments.

Select the Image Tile and File Descriptor for which you wish to make an assignment and then use the File Chooser to browse to the existing file you want to associate with the selected Descriptor.

Image Tile: Loudoun_TM

File Descriptor: Imported_Tile

File Chooser:

< Back
Next >
Close
Help

This popup list contains all the Image Tiles in the project

Select Classified Tile

Browse to the examples directory

The **Assign Tile Node Files** step allows you to select files that have already been prepared for the Sampling process and assign them to their proper places in the Project.

Tile Node Files

Imported Tile: An Imported Tile is a native IMAGINE format image that provides the initial synoptic view of the study area. This file provides the initial starting point for all of the files below. The inclusion of an Imported Tile in the project is optional, as long as you can provide a Stratified Tile.

Rectified Tile: A Rectified Tile is an orthorectified version of the Imported Tile. The Rectified Tile must undergo Classification to provide the Classified Tile below. The inclusion of a Rectified Tile in the project is optional, as long as you can provide a Stratified Tile.

For more information on orthorectification, see "Orthorectification" on page 265.

Classified Tile: A Classified Tile is a thematic classification of the Rectified Tile.

For instructions on classifying an image, refer to"Advanced Classification" on page 455.

Stratified Tile: The stratified tile is a refined grouping of the Classified Tile. This grouping is usually performed with the Class Grouping Tool. The Grouped image is then Recoded to include only those strata which contain the MOI. This file is **required** by the Frame Sampling process.

For tips and techniques on stratifying images, see "Using the Grouping Tool" on page 500.

Sampling Grid: The Sampling Grid contains the vector polygons needed for Sample Selection. The Sampling Grid is usually created with the **Grid Generation Tool**, but it can be a previously created Shapefile. This file is **required** by the Frame Sampling process.

Prior Data: The Prior Data file is any standard IMAGINE .img file that contains information about previous locations of the particular feature class of interest or variation of the occurrence of the material of interest within the study area. This information helps you choose which portions of the image to sample with high-resolution imagery. The inclusion of Prior Data in the project is optional.

Selected Samples: The Selected Samples file defines the Sampling Grid Cells that have been selected for analysis with high-resolution imagery. This file is usually created with the **Sample Selection Tool**, and is **required** by the Frame Sampling process.

Fraction File: The Fraction File is created by the Final Analysis process.

2. Select **Classified_Tile** from the File Descriptor popup list.

3. Click the Browse button and browse to the IMAGINE_HOME/examples directory. Select **loudoun_maxclass.img** from the list of files, and click **OK**.

4. Click **Next** on the Single Sampling Wizard.

   The **Create/Assign Stratum Files** step displays in the Wizard.

   The Grouped File we will use in this tour was created during the tour "Using the Grouping Tool" on page 500. If you have not already created the Grouped File, you can click the Class Grouping Tool button to launch the Grouping Tool.

Recoding the Grouped File

Now that you have Grouped the Classified File into class groups, it is necessary to recode the Grouped file so that only those classes which contain the MOI are included in the file. This will eliminate the possibility of MOI contribution from strata that have been designated as Non-MOI strata and will reduce the noise in the estimate. It will also increase the User Confidence in the Final Analysis.

1. From the **IMAGINE** icon panel, select the **Image Interpreter** icon.

   The Image Interpreter menu is displayed.

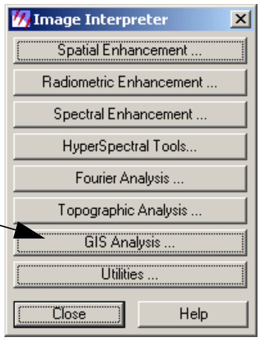
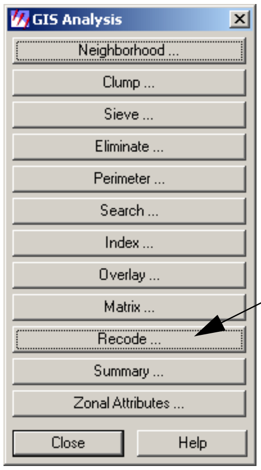


Image Interpreter
Spatial Enhancement ...
Radiometric Enhancement ...
Spectral Enhancement ...
HyperSpectral Tools...
Fourier Analysis ...
Topographic Analysis ...
GIS Analysis ...
Utilities ...
Close
Help

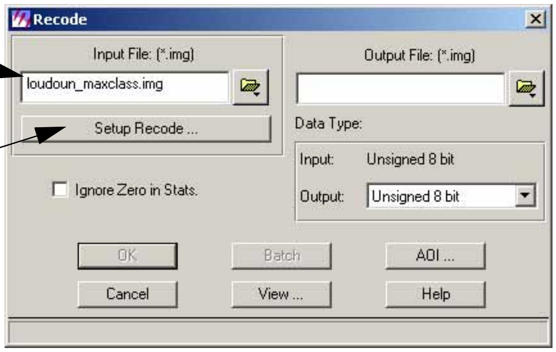
Click here to open
the GIS Analysis menu

2. Click the **GIS Analysis** button to open the **GIS Analysis** menu.


GIS Analysis
Neighborhood ...
Clump ...
Sieve ...
Eliminate ...
Perimeter ...
Search ...
Index ...
Overlay ...
Matrix ...
Recode ...
Summary ...
Zonal Attributes ...
Close
Help

Click here to open
the Recode utility.

3. Click the **Recode** button to open the **Recode** dialog.


Recode
Setup Recode ...
☐ Ignore Zero in Stats.
OK
Batch
AOI ...
Cancel
View ...
Help

Input File: (*.img) loudoun_maxclass.img

Output File: (*.img)

Data Type:

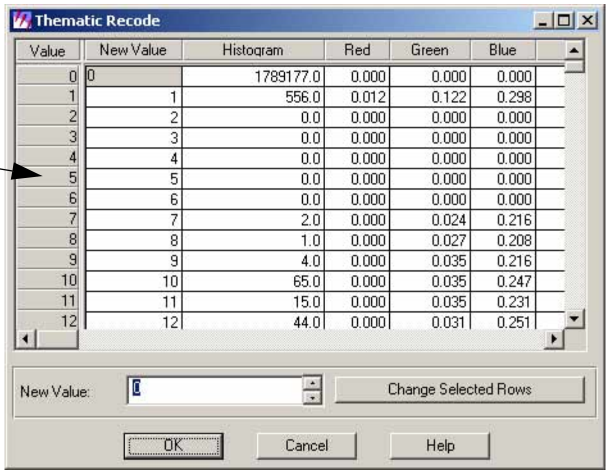
Input: Unsigned 8 bit

Output: Unsigned 8 bit

Enter the Grouped file here

Click to Setup the Recode

4. Click the **Setup Recode** button to open the **Thematic Recode** dialog.


Thematic Recode
Change Selected Rows
OK
Cancel
Help

Value	New Value	Histogram	Red	Green	Blue
0	0	1789177.0	0.000	0.000	0.000
1	1	556.0	0.012	0.122	0.298
2	2	0.0	0.000	0.000	0.000
3	3	0.0	0.000	0.000	0.000
4	4	0.0	0.000	0.000	0.000
5	5	0.0	0.000	0.000	0.000
6	6	0.0	0.000	0.000	0.000
7	7	2.0	0.000	0.024	0.216
8	8	1.0	0.000	0.027	0.208
9	9	4.0	0.000	0.035	0.216
10	10	65.0	0.000	0.035	0.247
11	11	15.0	0.000	0.035	0.231
12	12	44.0	0.000	0.031	0.251

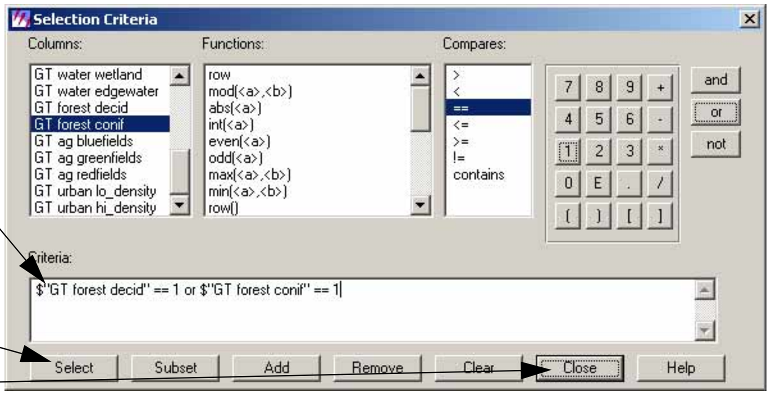
New Value: 0

Right-click here
and select Criteria...

Take a moment to look at the Columns that appear in the Thematic Recode CellArray. Notice that the columns that were created with the Grouping Tool are all labeled with GT TargetName GroupName. A 0 in this column means that the Class (Value column) is not included in this Group. A 1 indicates that Class is included in the Group.

5. Right-click in the Value column and select **Criteria..**. from the popup list.

   The Selection Criteria dialog is opened.


Selection Criteria
Columns:
GT water wetland
GT water edgewater
GT forest decid
GT forest conif
GT ag bluefields
GT ag greenfields
GT ag redfields
GT urban lo_density
GT urban hi_density
Functions:
row
mod(<a>,<b>)
abs(<a>)
int(<a>)
even(<a>)
odd(<a>)
max(<a>,<b>)
min(<a>,<b>)
row()
Compares:
>
<
==
<=
>=
!=
contains
7 8 9 +
4 5 6 -
1 2 3 *
0 E . /
( ) [ ]
and
or
not
Criteria:
Select
Subset
Add
Remove
Clear
Close
Help

Criteria: $"GT forest decid" == 1 or $"GT forest conif" == 1

To select the forested classes, set GT Forest columns == 1

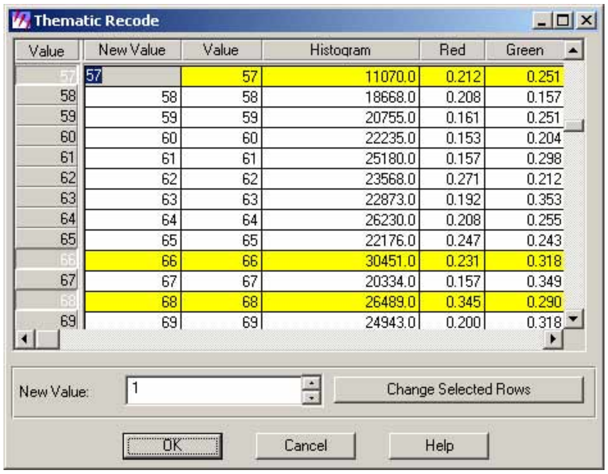
Click Select

Click Close

6. Set each of the **GT Forest** columns **== 1**.

7. Click the **Select** button.

   All of the columns that are grouped into the Forest Target Class should be selected in the **Thematic Recode** dialog.

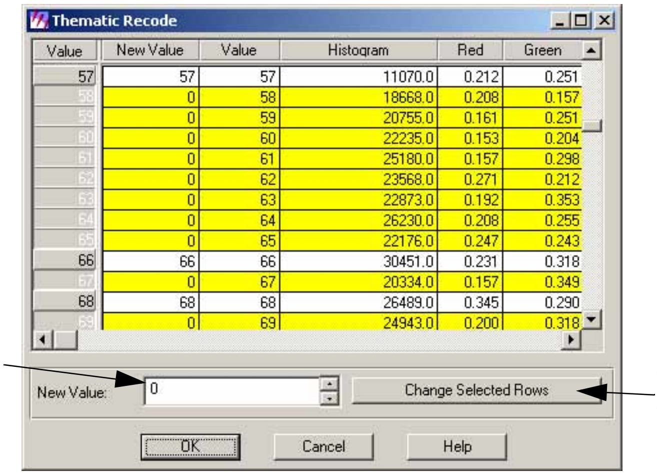

Thematic Recode
Change Selected Rows
OK
Cancel
Help

Value	New Value	Value	Histogram	Red	Green
57	57	57	11070.0	0.212	0.251
58	58	58	18668.0	0.208	0.157
59	59	59	20755.0	0.161	0.251
60	60	60	22235.0	0.153	0.204
61	61	61	25180.0	0.157	0.298
62	62	62	23568.0	0.271	0.212
63	63	63	22873.0	0.192	0.353
64	64	64	26230.0	0.208	0.255
65	65	65	22176.0	0.247	0.243
66	66	66	30451.0	0.231	0.318
67	67	67	20334.0	0.157	0.349
68	68	68	26489.0	0.345	0.290
69	69	69	24943.0	0.200	0.318

New Value: 1

8. Right-click in the Value column and select **Invert Selection**.

9. Enter **0** in the **New Value** field and click the **Change Selected Rows** button.



Value	New Value	Value	Histogram	Red	Green
57	57	57	11070.0	0.212	0.251
58	0	58	18668.0	0.208	0.157
59	0	59	20755.0	0.161	0.251
60	0	60	22235.0	0.153	0.204
61	0	61	25180.0	0.157	0.298
62	0	62	23568.0	0.271	0.212
63	0	63	22873.0	0.192	0.353
64	0	64	26230.0	0.208	0.255
65	0	65	22176.0	0.247	0.243
66	66	66	30451.0	0.231	0.318
67	0	67	20334.0	0.157	0.349
68	68	68	26489.0	0.345	0.290
69	0	69	24943.0	0.200	0.318

New Value: 0

Enter 0 here

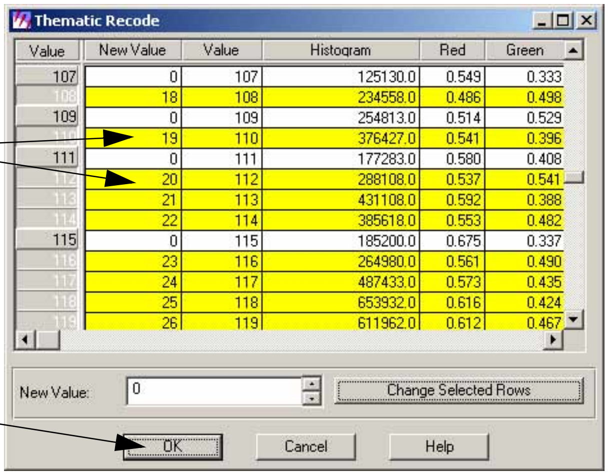
Click Change Selected Rows

All of the Classes that are not members of the Forest Target Class will have their pixel values set to 0. This will exclude them from the Stratum File and eliminate them from the computations of the Final Analysis.

10. Right-click on the Value column again and select **Invert Selection** from the popup list.

This selects only those classes that are members of the Forest Target Class.

11. Renumber the **New Values** for the members of the MOI Target class. The numbering system depends on your needs.



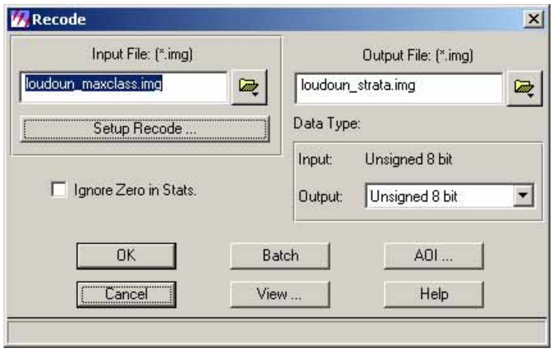
Value	New Value	Value	Histogram	Red	Green
107	0	107	125130.0	0.549	0.333
108	18	108	234558.0	0.486	0.498
109	0	109	254813.0	0.514	0.529
110	19	110	376427.0	0.541	0.396
111	0	111	177283.0	0.580	0.408
112	20	112	288108.0	0.537	0.541
113	21	113	431108.0	0.592	0.388
114	22	114	385618.0	0.553	0.482
115	0	115	185200.0	0.675	0.337
116	23	116	264980.0	0.561	0.490
117	24	117	487433.0	0.573	0.435
118	25	118	653932.0	0.616	0.424
119	26	119	611962.0	0.612	0.467

New Value: 0

Renumber the selected Classes

Click OK

12. Click **OK** to exit the **Thematic Recode** dialog and return to the **Recode** dialog.



Recode

Input File: (*.img) loudoun_maxclass.img

Output File: (*.img) loudoun_strata.img

Data Type:

Input: Unsigned 8 bit

Output: Unsigned 8 bit

☐ Ignore Zero in Stats.

Enter loudoun_strata.img here

13. In the Output Filename field, click the Browse button and browse to the project directory. Enter **loudoun_strata.img** as the File Name and click **OK** in the File Selector.

14. Click **OK** on the **Recode** dialog to start the Recode process. A Progress meter displays.

15. When the Progress meter reaches 100%, click **OK** to dismiss it.

NOTE: You may want to paste the color table from the grouped image to the Attribute Editor of the new stratum file.Use the same criteria selection method as described above to copy the MOI colors to the Stratum file.

Generate the Sampling Grid

1. In the **Create/Assign Stratum File** step, click the Browse button and browse to the directory in which you created the stratum file. Select **loudoun_strata.img** from the list of files and click **OK**.

2. Click **Next** in the Wizard.

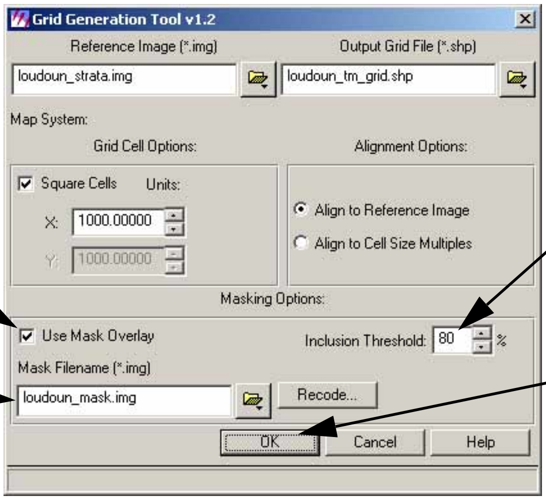
   The **Create/Assign Sampling Grid** step is displayed.

3. In the **Create/Assign Sampling Grid** step, click the Create Sampling Grid button to create a Sampling Grid file.

   The **Grid Generation Tool** is opened.

Using the Grid Generation Tool

The Grid Generation Tool is used to create a Shapefile grid that will overlay the Stratum file.



Reference Image (*.img): loudoun_strata.img

Output Grid File (*.shp): loudoun_tm_grid.shp

Map System:

Grid Cell Options:

☑ Square Cells Units:

X: 1000.00000

Y: 1000.00000

Alignment Options:

● Align to Reference Image

○ Align to Cell Size Multiples

Masking Options:

☑ Use Mask Overlay

Inclusion Threshold: 80 %

Mask Filename (*.img): loudoun_mask.img

Select the
Use Mask Overlay
checkbox

Browse to
loudoun_mask.img

Enter 80% as the
Inclusion Threshold

Click OK to generate
the Shapefile

1. Make sure that the Reference Image is set as **loudoun_strata.img**.

2. Note that the Output Grid File is **loudoun_tm_grid.shp**. This is the default name, which has the appends _grid.shp to the Tile Name. If you change the Output File Name, that change will be reflected in the Project Manager.

3. Select the **Use Mask Overlay** checkbox.

   The Mask file is used to limit the coverage of the Sampling Grid. Because the Sampling that can be performed in this tour is restricted by our ability to use only the two existing high resolution files, we use the mask file to mask out all of the portions of the Tile that do not have high resolution imagery coverage.

4. Next to the Mask Filename part, click the Browse button and browse to the IMAGINE_HOME/examples directory. Select **loudoun_mask.img** from the list of files and click **OK**.

5. Enter **80** as the Inclusion Threshold.

   Setting the Inclusion Threshold to 80 will ensure that at least 80% of every Sample Cell created by the Grid Generation Tool falls within the bounds set by the Mask File.

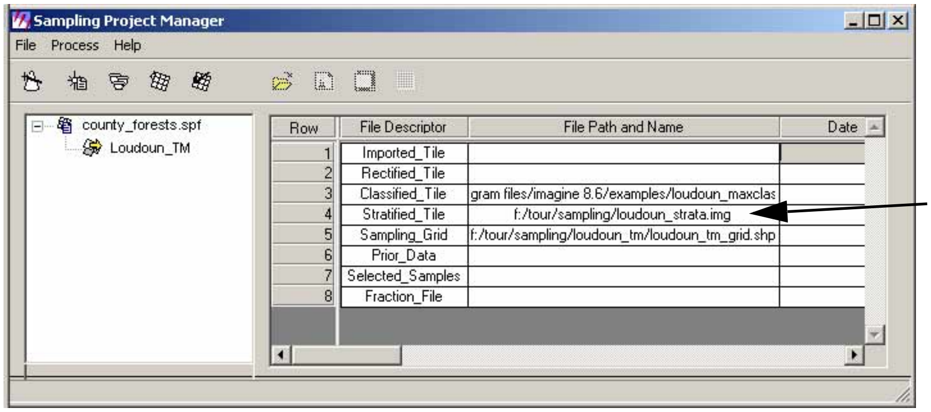
6. Click **OK** to create the Sampling Grid and return to the Sampling Project Wizard.

   A Progress meter appears and tracks the progress of the Grid Generation Process.

7. When the **Create/Assign Sampling Grid** step reappears, click **Next**.

   The **Select Samples** step is displayed.

   Note that the Project Manager is updated to include the Sampling Grid file you just created.



Row	File Descriptor	File Path and Name	Date
1	Imported_Tile		
2	Rectified_Tile		
3	Classified_Tile	gram files/imagine 8.6/examples/loudoun_maxclas	
4	Stratified_Tile	f:/tour/sampling/loudoun_strata.img	
5	Sampling_Grid	f:/tour/sampling/loudoun_tm/loudoun_tm_grid.shp	
6	Prior_Data		
7	Selected_Samples		
8	Fraction_File		

The new sampling grid file was placed in a directory named loudoun_tm

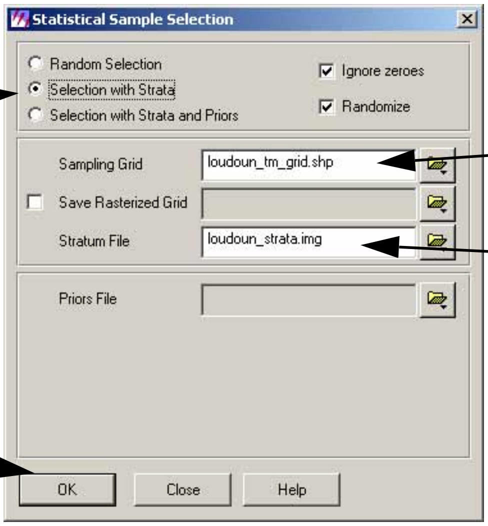
All of the Tile Node Level files are created in a Tile Level directory which bears the same name as the Tile itself.

Selecting the Samples

The Statistical Sample Selection dialog allows you to choose which files you would like to open to aid you in the selection of the Sample Cells which will be interpreted for the MOI.

1. In the Select Samples step of the Sampling Wizard, click the Select Samples button .

   The Statistical Sample Selection dialog displays.



○ Random Selection
● Selection with Strata
○ Selection with Strata and Priors
☑ Ignore zeroes
☑ Randomize

Sampling Grid: loudoun_tm_grid.shp
☐ Save Rasterized Grid:
Stratum File: loudoun_strata.img
Priors File:

Select the Selection with Strata radio button

The Sampling Grid file displays here

Browse to the Stratum File created in the Recoding the Grouped File step

Click OK to open the Sample Selection Tool

2. Select the Selection with Strata radio button.

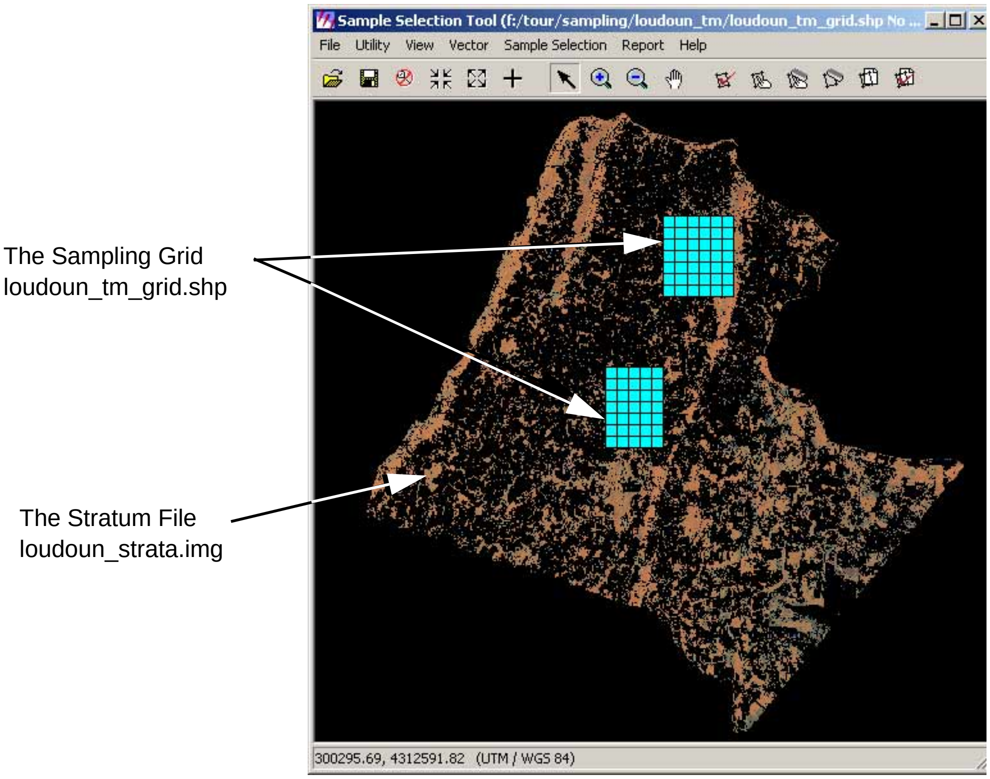
3. Make sure that the Sampling Grid filename part displays the **loudoun_tm_grid.shp** file.

4. Make sure that the Stratum filename part displays the **loudoun_strata.img** file. If it does not, click the Browse button and browse to the file. Select **loudoun_strata.img** from the list of files and click **OK**.

5. Click **OK** in the Statistical Sample Selection dialog to open the **Sample Selection Tool**.

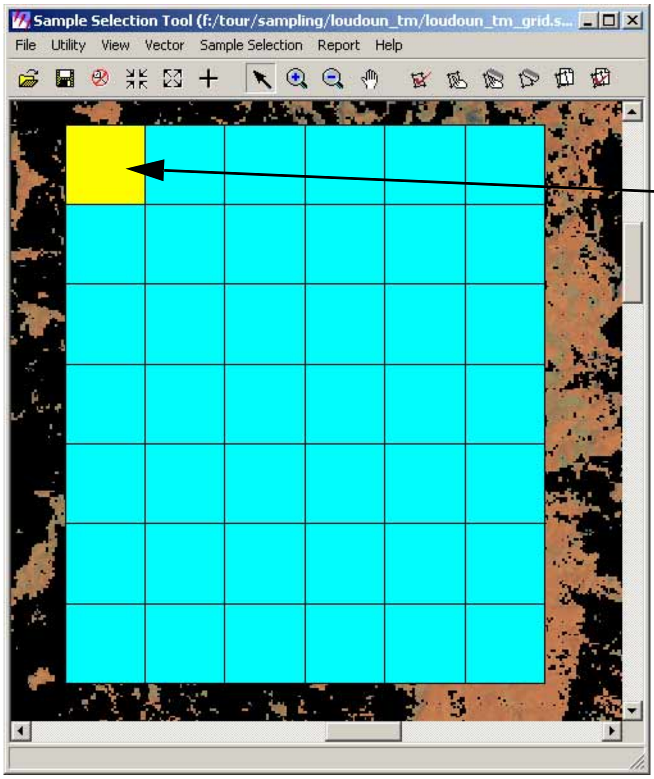
   The Sample Selection Tool is opened

   .


Sample Selection Tool (f:/tour/sampling/loudoun_tm/loudoun_tm_grid.shp No ...
File Utility View Vector Sample Selection Report Help
The Sampling Grid
loudoun_tm_grid.shp
The Stratum File
loudoun_strata.img
300295.69, 4312591.82 (UTM / WGS 84)

Manually Selecting Cells

1. Using the Manual Zoom tool zoom in on the upper portion of the Sampling Grid.


Sample Selection Tool (f:/tour/sampling/loudoun_tm/loudoun_tm_grid.s...
File Utility View Vector Sample Selection Report Help

Click here to highlight this Cell

2. Click on the Selector icon in the Sample Selection toolbar.
3. Click on the indicated Sample Cell to highlight it.
4. Select **Utility | Blend** from the Sample Selection menu bar.

   The **Viewer Blend / Fade** dialog is opened.
5. Use the meter handle to adjust the amount of blending so that you can view the Stratum File through the Grid. This allows you to select Sample Cells that contain a representative amount to the MOI.

   You must exercise caution when manually selecting Sample Cells for interpretation. No more than half of the Samples should be manually selected. Manually selecting more Cells will introduce user bias into the calculations.
6. When you have finished viewing the Stratum file, click **OK** on the **Viewer Blend/Fade** dialog to dismiss it.

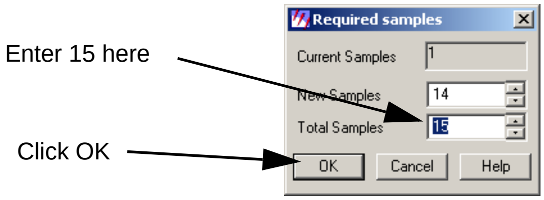
7. Click the Accept Manually Selected Cells button to select this Cell for interpretation.

Automatically Selecting Sample Cells

The Sample Selection Tool provides a utility for randomly selecting Cells for interpretation. This utility automatically selects cells based on the size and expected proportion of the MOI in the stratum.

1. Click the Automatic Selection icon in the Sample Selection toolbar.

   The Required Samples dialog is opened.



Current Samples 1

New Samples 14

Total Samples 15

Enter 15 here

Click OK

   Note that the Current Samples number box displays the number 1. This is the Cell that we manually selected.

2. In the Total Samples number box, enter **15** and press Enter.

   The New Samples number will update to 14, indicating the number of samples that the program needs to automatically identify.

3. Click **OK** on the **Required Samples** dialog to close it and automatically select 14 additional Cells.

   NOTE: Because the Automatic Selection process is random, the automatically selected samples may differ from those in this tour.

4. Select **File | Save Selected As...** to save the selected Cells as a new shapefile.

   The **Save Sampling Grid As:** dialog is opened.

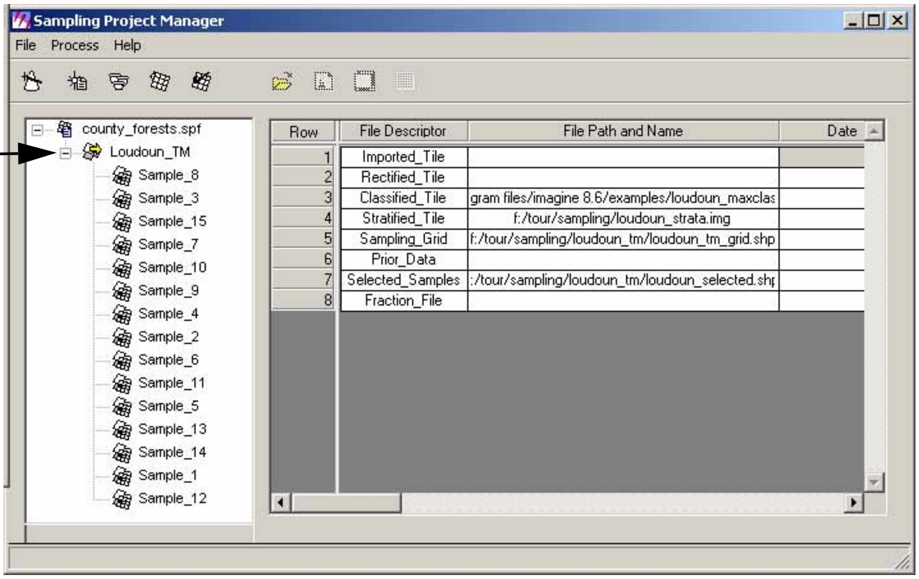
5. Navigate to the **loudoun_tm** directory that contains the **loudoun_tm_grid.shp** file.

6. Enter **loudoun_selected.shp** in the filename and click **OK** to save the shapefile.

7. Select **File | Close** to dismiss the **Sample Selection Tool** and return to the **Sampling Project Wizard**.

   A Progress meter displays as the Sample Selection Tool creates AOI bounding boxes for each of the selected Samples.

   Note that the Project Manager has been updated to include the 15 samples we have selected.



Click here to expand the Tile level node

- county_forests.spf
  - Loudoun_TM
    - Sample_8
    - Sample_3
    - Sample_15
    - Sample_7
    - Sample_10
    - Sample_9
    - Sample_4
    - Sample_2
    - Sample_6
    - Sample_11
    - Sample_5
    - Sample_13
    - Sample_14
    - Sample_1
    - Sample_12

Row	File Descriptor	File Path and Name	Date
1	Imported_Tile		
2	Rectified_Tile		
3	Classified_Tile	gram files/imagine 8.6/examples/loudoun_maxclas	
4	Stratified_Tile	f:/tour/sampling/loudoun_strata.img	
5	Sampling_Grid	f:/tour/sampling/loudoun_tm/loudoun_tm_grid.shp	
6	Prior_Data		
7	Selected_Samples	:/tour/sampling/loudoun_tm/loudoun_selected.sh	
8	Fraction_File		

Sample Level Functions

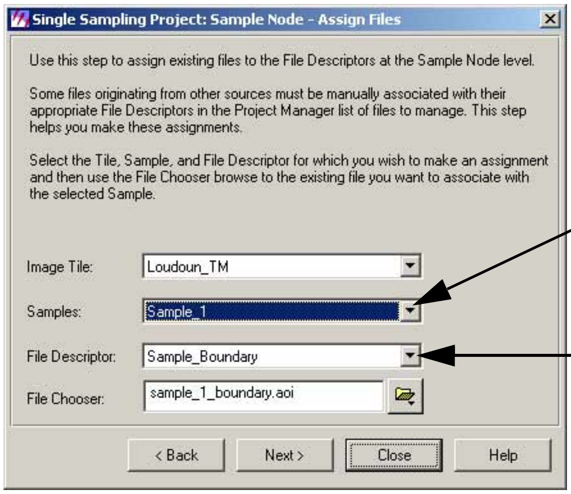
We are now ready to perform the Sample Level functions.

1. Click **Next** on the Select Samples step of the Sampling Wizard.

   The **Set Up Sample Node Level Files** step is displayed.

2. Click **Next** on the **Setup Sample Node Files** step of the Sampling Wizard.

   The **Assign Sample Node Files** step is displayed.



Use this step to assign existing files to the File Descriptors at the Sample Node level.

Some files originating from other sources must be manually associated with their appropriate File Descriptors in the Project Manager list of files to manage. This step helps you make these assignments.

Select the Tile, Sample, and File Descriptor for which you wish to make an assignment and then use the File Chooser browse to the existing file you want to associate with the selected Sample.

Image Tile: Loudoun_TM
Samples: Sample_1
File Descriptor: Sample_Boundary
File Chooser: sample_1_boundary.aoi

This popup list contains all of the samples for the selected Tile

This list contains all of the file descriptors for the selected Sample

This dialog is used to assign files that are associated with each of the Samples to their respective Sample Cell.

Sample Node Files

Sample Boundary: The Sample Boundary is a polygon AOI file that traces the boundary of the Selected Cell. This file is automatically created and by the Project Manager after Sample Selection. It is a required file.

Imported Sample: An Imported Sample is a native IMAGINE format image that provides high resolution view of the representative sample. The inclusion of an Imported Sample in the project is optional, as long as you can provide a Rectified Sample.

Rectified Sample: A Rectified Sample is an orthorectified version of the Imported Sample. The Rectified Sample will be used to perform the high resolution interpretation of the Sample Cells. This file is required by the Sampling Process.

The Rectified Sample files must be manually assigned to the appropriate samples.

Dot Grid Interpretation: The Dot Grid Interpretation is an annotation file (.ovr) that is the result of a Dot Grid Interpretation of the high resolution sample. This file created by the Dot Grid Tool. This file is required for this sample to be included Dot Grid Final Analysis.

Polygon Interpretation: The Polygon Interpretation is the output file of a Polygon Interpretation of the high resolution sample. This file is created by the Polygon Analysis Tool. This file is required for this sample to be included Polygon Final Analysis.

Assigning the Rectified Samples

The first step in assigning the Rectified Samples is to find out which selected cells overlap which high resolution image.

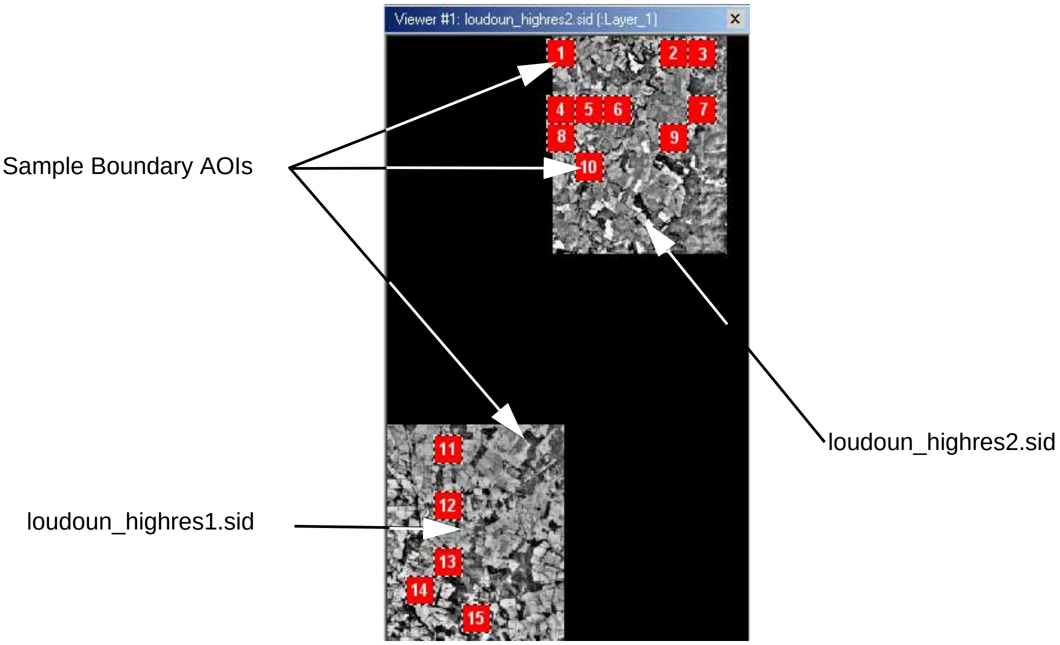
1. On the ERDAS IMAGINE icon panel, click the Viewer icon to open a **Viewer**.
2. Click the Open file icon on the **Viewer** toolbar.

   The **File Chooser** is opened.
3. Browse to the **IMAGINE_HOME/examples** directory.
4. From the Files of Type popup list, select **MrSID (*.sid)**.
5. Ctrl-click **loudoun_highres1.sid** and **loudoun_highres2.sid**.
6. Click **OK** to open the files in the **Viewer**.
7. Click the Open file icon on the **Viewer** toolbar.

   The **File Chooser** is opened.
8. Browse to the directory that contains your Sampling Project.

9. Select the **loudoun_tm/sample_1** directory.
10. From the Files of Type popup list, select **AOI (*.aoi)**.
11. Select **sample_1_boundary.aoi** and click **OK**.

    The AOI that defines the boundary of **Sample_1** is opened in the Viewer.


Viewer #1: loudoun_highres2.sid (:Layer_1)
Sample Boundary AOIs
1
2
3
4
5
6
7
8
9
10
loudoun_highres2.sid
11
12
13
14
15
loudoun_highres1.sid

12. In the Assign Files step of the **Sampling Wizard**, select **Sample_1** from the Samples popup list.
13. Select **Rectified_Sample** from the File Descriptor popup list.
14. Click the Browse icon and browse to the **IMAGINE _HOME/examples** directory.
15. In the Files of Type popup list, select **MrSID (*.sid)**.
16. Select the high resolution file that overlaps **Sample_1** (**loudoun_highres2.sid**) and click **OK**.
17. Repeat steps (7.) through (16.) for each of the Samples.

Dot Grid Interpretation

Dot Grid Interpretation will overlay a grid of dots on the portion of the high-resolution image contained within the Sample. You will label these dots so that they correctly identify the underlying features. The labeled grid is used to calculate the percentage of the MOI occurring within that portion of the Stratum File.

Placing the Dot Grid

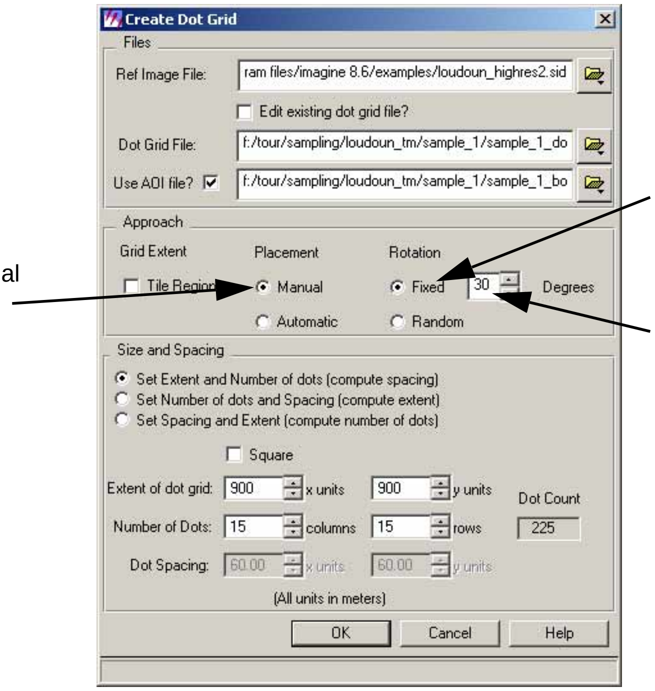
1. Click **Next** on the Sampling Wizard.

   The Interpret High Resolution Samples step is opened.

2. In the Sample popup list, select **Sample_1**.

3. Click the Perform Dot Grid Interpretation icon to open the **Dot Grid Tool**.

   The Create Dot Grid dialog is opened.



Create Dot Grid

Files

Ref Image File: ram files/imagine 8.6/examples/loudoun_highres2.sid

☐ Edit existing dot grid file?

Dot Grid File: f:/tour/sampling/loudoun_tm/sample_1/sample_1_do

Use AOI file? ☑ f:/tour/sampling/loudoun_tm/sample_1/sample_1_bo

Approach

Grid Extent: ☐ Tile Region

Placement: ● Manual ○ Automatic

Rotation: ● Fixed 30 Degrees ○ Random

Size and Spacing

● Set Extent and Number of dots (compute spacing)
○ Set Number of dots and Spacing (compute extent)
○ Set Spacing and Extent (compute number of dots)

☐ Square

Extent of dot grid: 900 x units 900 y units

Number of Dots: 15 columns 15 rows

Dot Spacing: 60.00 x units 60.00 y units

Dot Count 225

(All units in meters)

Select Manual Placement

Select Fixed Rotation

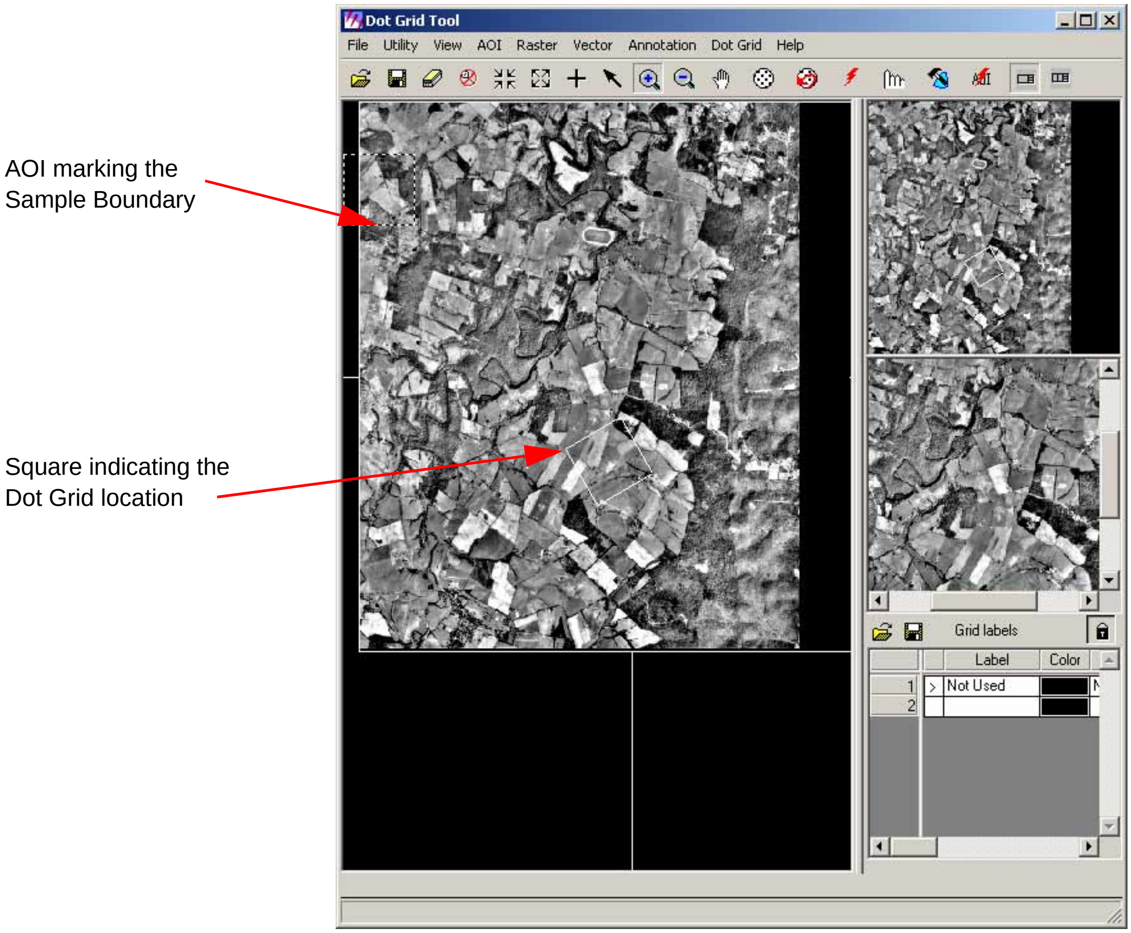
Enter 30 here

4. In the Approach group, select the **Manual Placement** radio button. This will have the program randomly place the Dot Grid within the Sample.

5. Select the **Fixed Rotation** radio button. Enter **30** in the Fixed Rotation number field.

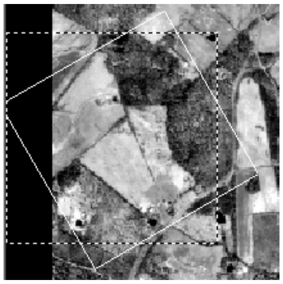
6. Click **OK**.

   The high resolution image displays in the **Dot Grid Tool**. An AOI is placed in the image to demarcate the boundary of the Sample. A square indicates the location of the Dot Grid.


AOI marking the Sample Boundary
Square indicating the Dot Grid location
Dot Grid Tool
File Utility View AOI Raster Vector Annotation Dot Grid Help
Grid labels

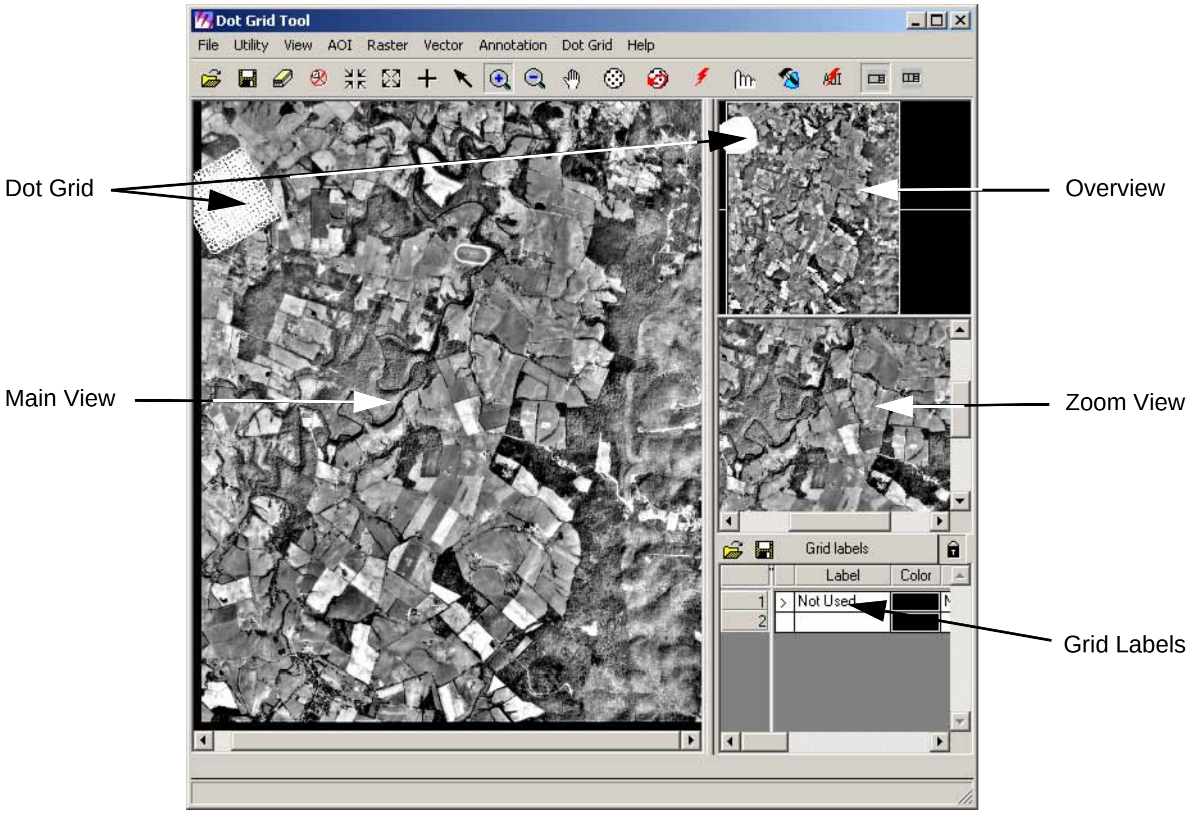
	Label	Color
1	Not Used	■
2		■

7. Drag the Dot Grid square until it covers the majority of the Sample.



8. Double-click inside the square to create the Dot Grid.

Dot Grid Tool


Dot Grid Tool
File Utility View AOI Raster Vector Annotation Dot Grid Help
Dot Grid
Overview
Main View
Zoom View
Grid labels
Label
Color
1
Not Used
2
Grid Labels

Set Up the Grid Labels

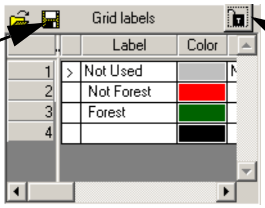
The first step in the Interpretation of the Samples is to create a Label set that will be used to label all of the Samples.

1. Determine the number of labels that you will use to interpret the Sample. In this tour, we will use three labels: **Forest**, **Not Forest**, and **Not Used**.
2. Select the **Append new row to Grid Label CellArray** item from the **Dot Grid** menu. Repeat this for every label you will add to the Label Set.
3. In the Grid Labels Group, click on the Locked icon to enable the editing of the Grid Labels.

   The icon changes to indicate that the labels have been unlocked .
4. Click **OK** on the message that informs you that this label set will be applied to all of the samples in the project.
5. Click in the Label Column of Row 2. Type **Not Forest**.
6. Click in the Label Column for Row 3. Type **Forest**.

7. Click in the Color column for the **Not Used** label. Select **Gray** from the list of colors that appears.
8. Click in the Color column for the **Not Forest** label. Select **Red** from the list of colors that appears.
9. Click in the Color column for the **Forest** label. Select **Dark Green** from the list of colors that appears.

   The Grid Labels will look like this:



	Label	Color
1	Not Used	▒
2	Not Forest	🟥
3	Forest	🟩
4		⬛

Click here to Save the Grid Labels

Click to Lock the Grid Labels

10. Click the Save icon to Save the current label set.
11. Read the **Warning Message**. Click **Save Label Set** on the Warning Message.

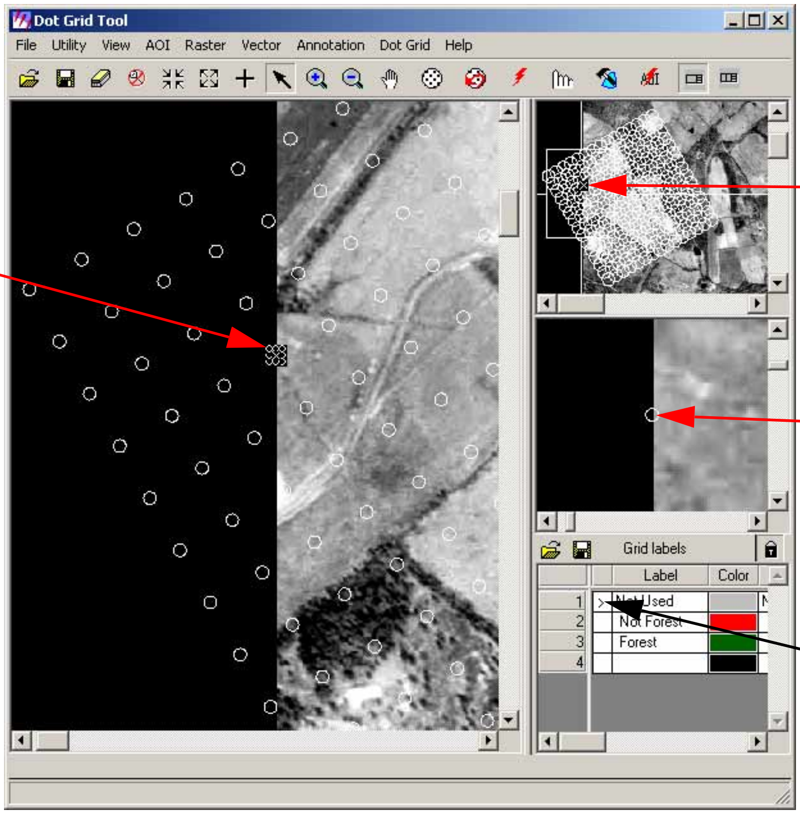
    The **File Chooser** opens.
12. Browse to the Sampling Project directory.

It is generally a good idea to save the Label Set in the Tile level directory. This keeps the Labels in a logical place within the project files hierarchy.

13. Enter **forest_moi_labels.lbs** as the filename.
14. Click **OK** to save the Label Set.

Manually Label the Grid

1. Use the Manual Zoom tool 🔍 to zoom in on the portion of the Dot grid that falls outside of the high resolution image.
2. Use the Manual Zoom tool 🔍 in the Zoom View to zoom in to a comfortable magnification.
3. Use the Manual Zoom tool 🔍 in the Overview so that it displays the extent of the Dot Grid.
4. Click the Select icon ↖ on the Dot Grid Tool toolbar.
5. Select a dot on the edge of the image by clicking on it in the Main View.


Dot Grid Tool
File Utility View AOI Raster Vector Annotation Dot Grid Help
Grid labels
Label
Color
1 > Not Used
2 Not Forest
3 Forest
4

Click on this dot to select it

The dot is also selected in the overview

The dot is centered in the Zoom View

Click here to set the Not Used label as the current label

The Zoom View shows that over half of the dot is outside of the image.

6. Set the carat > in the **Not Used** row of the Grid Labels by clicking in the first column.

7. Label the selected dot by clicking the Label Selection icon.

   The dot is filled with the color of the current label (Not Used) to indicate that it has been labeled.

Automatically Apply Labels

1. Click the Manual Label icon to toggle on the Automatic Label mode. The Automatic Label icon indicates that Automatic Label Mode is active.
2. Make sure that the **Not Used** label is still set as the current label in the Labels CellArray.
3. Select another of the dots that lies outside of the high resolution image extent.

   The **Not Used** label is automatically applied to the dot as it is selected.

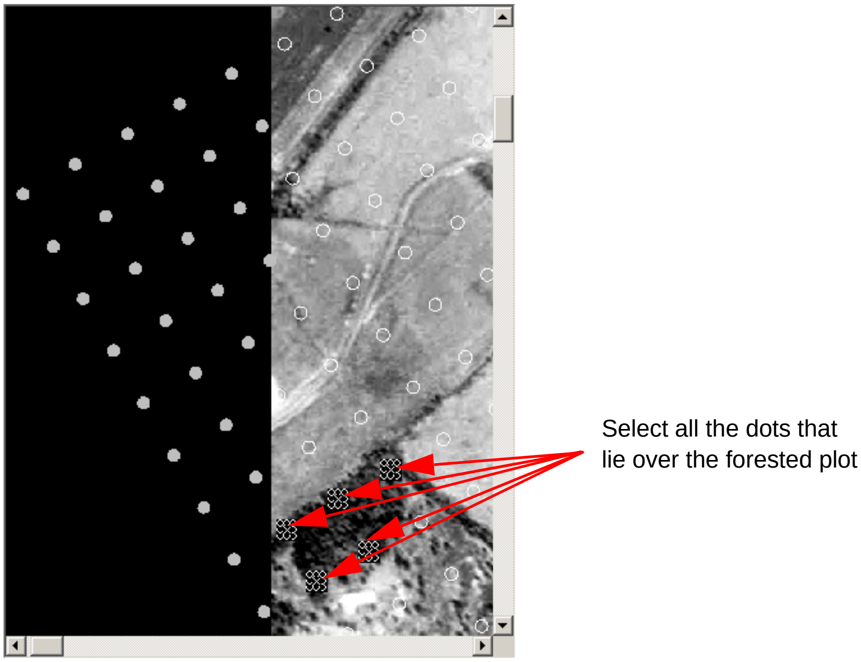
4. Repeat this process to label the dots outside of the image extent.

Label Multiple Dots

1. Click the Automatic Label icon to toggle on the Manual Label mode. The Manual Label icon indicates that Manual Label Mode is active.

   In the lower portion of the Main View, there is an area covered by trees.

2. Click on one of the dots in this portion of the image to select it.
3. Shift-click on the other dots that overlay this forested plot.



Select all the dots that lie over the forested plot

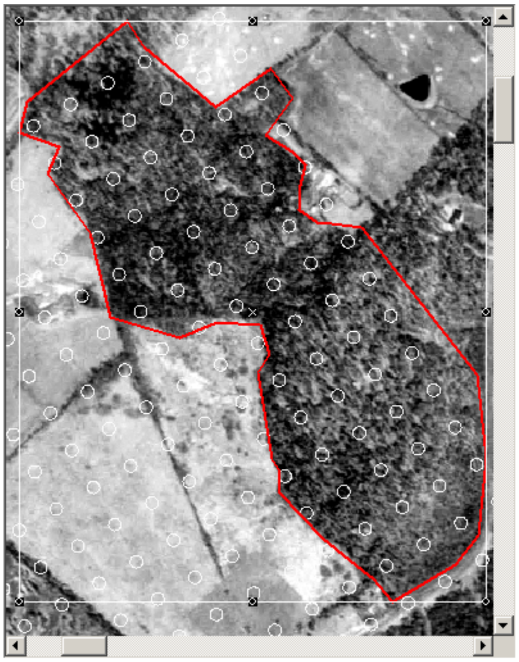
4. Place the carat > in the **Forest** label row to set it as the current label.
5. Label the selected dots by clicking the Label Selection icon .

Use AOI to Label Multiple Dots

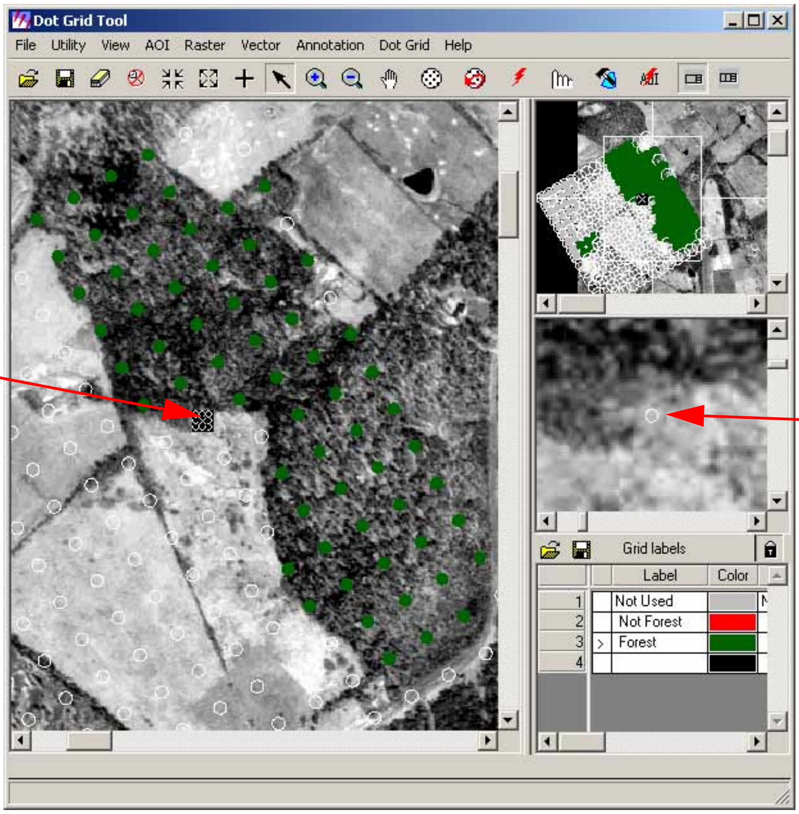
1. Use the Manual Zoom tool to zoom in on the large forested plot in the upper-left portion of the Dot Grid.
2. Open the **AOI Tools** by clicking on the AOI Tools icon on the **Dot Grid** toolbar.

The **AOI Tool Palette** is opened.

3. Select the Create Polygon AOI icon .
4. Digitize a polygon around the forested portion of the image.



5. In the Label CellArray, place the carat > in the **Forest** label.
6. Click the Label AOI icon to label all of the dots within the polygon.
7. Remove the AOI by selecting **AOI | Cut**.
8. Select a dot that lies along the perimeter of the polygon.


Dot Grid Tool
File Utility View AOI Raster Vector Annotation Dot Grid Help
Grid labels

	Label	Color
1	Not Used	▨
2	Not Forest	🟥
3	> Forest	🟩
4		■

Select this dot

The Zoom View shows it is actually Not Forest

Use the Zoom View to analyze whether or not the selected dot is correctly labeled.

9. If it is incorrectly labeled, place the carat > in the **Not Forest** label row, and click the Label Selected Dot icon ⚡.

10. Continue to analyze the dots that lie along the perimeter of the polygon, relabeling those that were erroneously included in the polygon.

Continue Interpretation

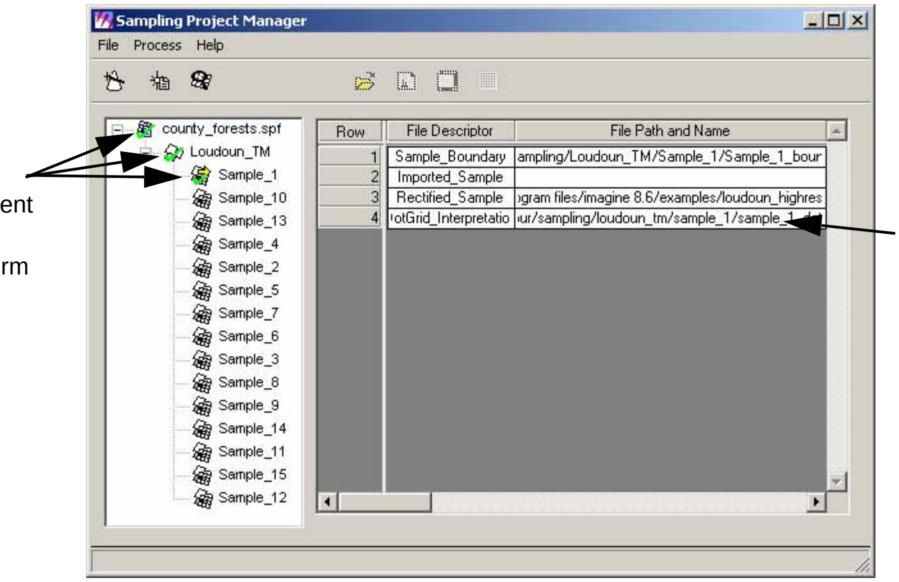
1. Continue labeling the Dot Grid until the entire Grid is correctly labeled.

2. Select **File | Save | Save Dot Grid** from the **Dot Grid Tool** menu bar.

3. Click **File | Quit** to exit the **Dot Grid Tool**.

   The **Dot Grid Tool** closes and you are returned to the **Interpret Samples** step of the Wizard.

   The **Project Manager** is updated to include the new Interpretation file.


Sampling Project Manager
File
Process
Help
county_forests.spf
Loudoun_TM
Sample_1
Sample_10
Sample_13
Sample_4
Sample_2
Sample_5
Sample_7
Sample_6
Sample_3
Sample_8
Sample_9
Sample_14
Sample_11
Sample_15
Sample_12

Row	File Descriptor	File Path and Name
1	Sample_Boundary	ampling/Loudoun_TM/Sample_1/Sample_1_bour
2	Imported_Sample	
3	Rectified_Sample	gram files/imagine 8.6/examples/loudoun_highres
4	otGrid_Interpretatio	ur/sampling/loudoun_tm/sample_1/sample_1

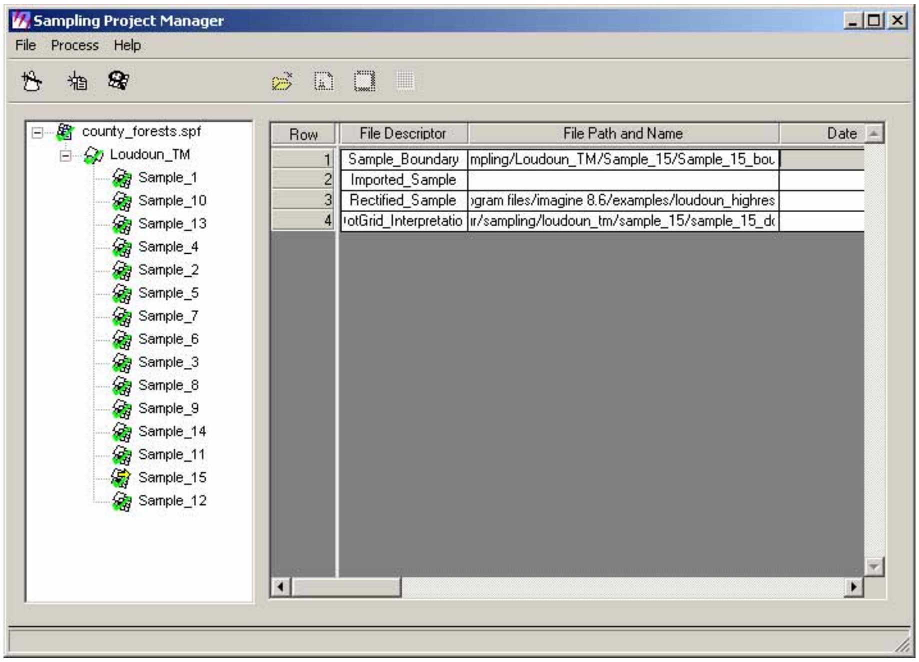
The green checks indicate that sufficient steps have been completed to perform Final Analysis

The Dot Grid file is added to the Project Manger

Although a Final Analysis can be run at this point, the accuracy of the analysis will be affected by the limited number of samples that have been analyzed.

4. Continue to interpret the Samples until they have all been labeled. Experiment with the Size and Spacing of the Dot Grid, as well as the Automatic Placement and Rotation options.

The Project Manager will indicate that all of the Samples have been interpreted and Final Analysis may be performed by placing green check marks in the Tree View.



- county_forests.spf
  - Loudoun_TM
    - Sample_1
    - Sample_10
    - Sample_13
    - Sample_4
    - Sample_2
    - Sample_5
    - Sample_7
    - Sample_6
    - Sample_3
    - Sample_8
    - Sample_9
    - Sample_14
    - Sample_11
    - Sample_15
    - Sample_12

Row	File Descriptor	File Path and Name	Date
1	Sample_Boundary	mpling/Loudoun_TM/Sample_15/Sample_15_bou	
2	Imported_Sample		
3	Rectified_Sample	gram files/imagine 8.6/examples/loudoun_highres	
4	otGrid_Interpretatio	r/sampling/loudoun_tm/sample_15/sample_15_d	

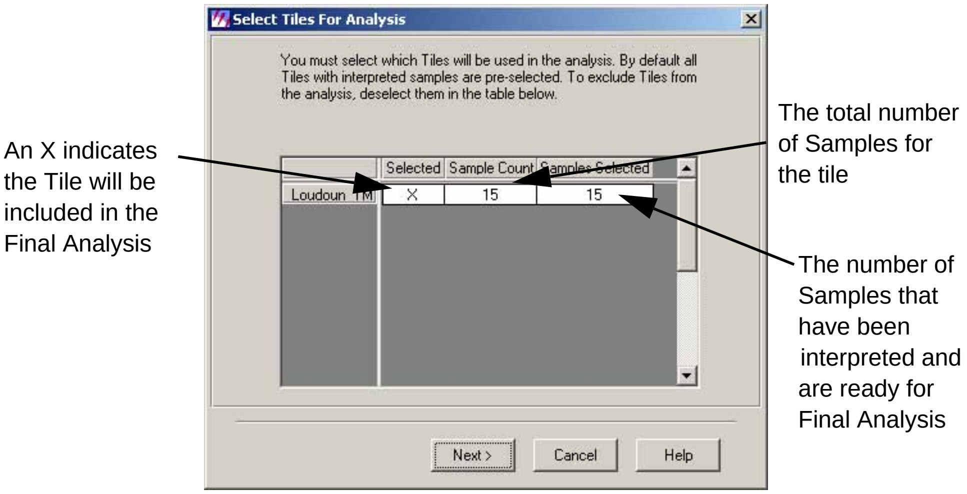
5. Click **Next** in the Interpret Samples step of the **Sampling Project Wizard**. The Final Analysis step is displayed.

6. Click the Final Analysis icon to start the **Final Analysis Wizard**.

Final Analysis Wizard

The **Final Analysis Wizard** lets you set the parameters which will dictate how the Final Analysis process will run.

The Final Analysis Wizard opens with the Select Tiles for Analysis step displayed.



Select Tiles For Analysis

You must select which Tiles will be used in the analysis. By default all Tiles with interpreted samples are pre-selected. To exclude Tiles from the analysis, deselect them in the table below.

	Selected	Sample Count	Samples Selected
Loudoun TM	X	15	15

An X indicates the Tile will be included in the Final Analysis

The total number of Samples for the tile

The number of Samples that have been interpreted and are ready for Final Analysis

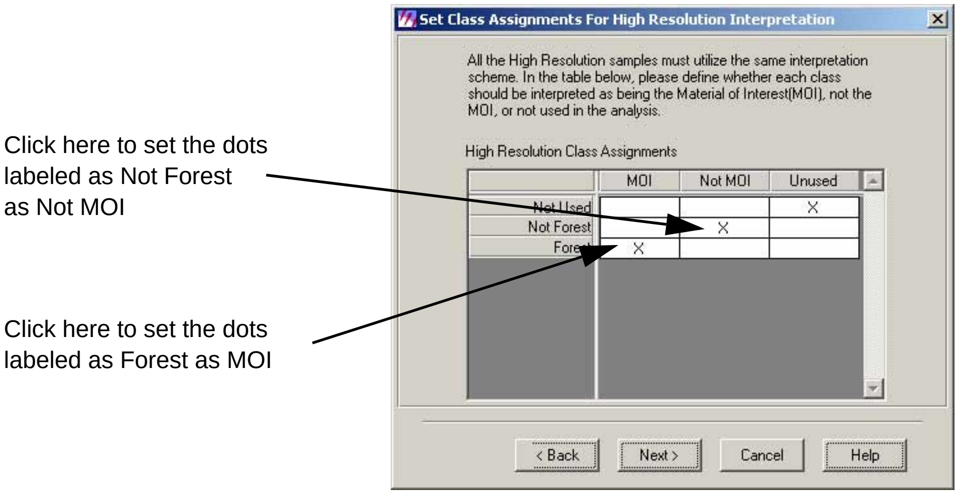
NOTE: If you enabled both Dot Grid and Polygon Analysis when you created the Sampling Project, a preliminary step will display asking you to choose which sampling method to use in the calculations.

1. The current Sampling project only uses one Image Tile, so click **Next** on the wizard.

   The Select Samples To Be Used step is displayed. If any of the Samples did not represent a good sampling of the MOI (e.g., it was centered over a lake or desert) you could exclude that sample from the Analysis.

2. Leave all of the Samples selected and click **Next** on the wizard.

   The Set Class Assignments For High Resolution Interpretation step is displayed.



Set Class Assignments For High Resolution Interpretation

All the High Resolution samples must utilize the same interpretation scheme. In the table below, please define whether each class should be interpreted as being the Material of Interest(MOI), not the MOI, or not used in the analysis.

High Resolution Class Assignments

	MOI	Not MOI	Unused
Not Used			X
Not Forest		X	
Forest	X		

Click here to set the dots labeled as Not Forest as Not MOI

Click here to set the dots labeled as Forest as MOI

3. Click in the Not Forest row, Not MOI column to set all of the dots in the Dot Grid that were labeled as Not Forest to be Not the Material of Interest in the Final Analysis.

4. Click in the Forest row, MOI column to set all of the dots that were labeled as Forest to be the Material of Interest for Final Analysis.

5. Leave the **X** in the Not Used row, Unused column to exclude these dots from the Final Analysis computations.

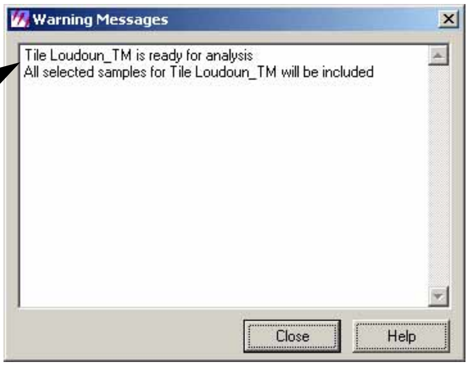
6. Click **Next** in the Final Analysis Wizard.

   The Check File Integrity step is displayed.

7. Click **Next** on the Wizard to perform with the project integrity check.

   The Final Analysis process will perform some preliminary checks to make sure that a Final Analysis can be performed.

8. Click on the **View Warnings** button. The **Warning Messages** dialog is displayed.



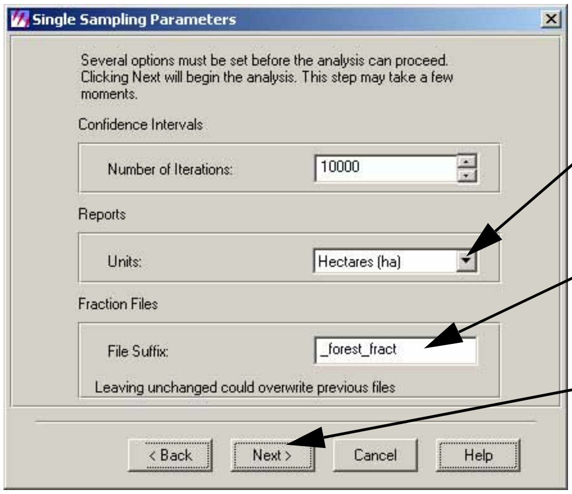
Tile Loudoun_TM is ready for analysis
All selected samples for Tile Loudoun_TM will be included

Any problem that prohibits a Final Analysis will be displayed at the top of this list of messages

9. Click **Close** to exit the **Warning Messages** dialog.

10. Click **Next** in the **Final Analysis Wizard**.

    The Single Sampling Parameters step is displayed.



Several options must be set before the analysis can proceed. Clicking Next will begin the analysis. This step may take a few moments.

Number of Iterations: 10000

Units: Hectares (ha)

File Suffix: _forest_fract

Leaving unchanged could overwrite previous files

Select Hectares from this list

Enter **_forest_fract** as the File Suffix

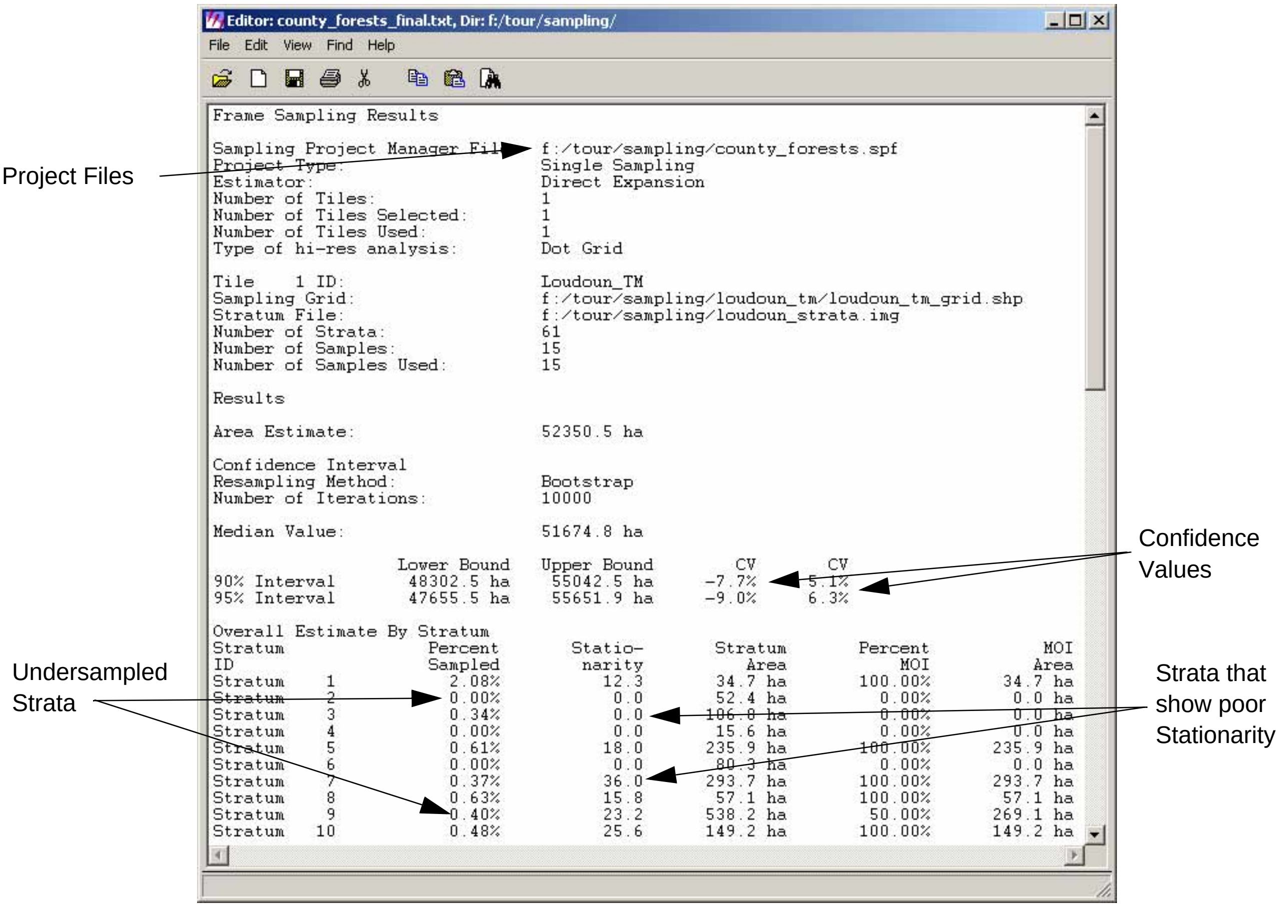
Click Next

11. Select **Hectares** from the Units popup list as the units in which to perform all the calculations.

12. Enter **_forest_fract** as the suffix for the Fraction File. This will identify the MOI for this fraction file in the Project Manager.

13. Click **Next** in the Final Analysis Wizard.

    The View Analysis Results step displays in the **Wizard**. The **Final Analysis Report** is opened in a **Text Editor** window.


Editor: county_forests_final.txt, Dir: f:/tour/sampling/
File
Edit
View
Find
Help
Project Files
Confidence Values
Undersampled Strata
Strata that show poor Stationarity

```
Frame Sampling Results

Sampling Project Manager Fil     f:/tour/sampling/county_forests.spf
Project Type:                    Single Sampling
Estimator:                       Direct Expansion
Number of Tiles:                 1
Number of Tiles Selected:        1
Number of Tiles Used:            1
Type of hi-res analysis:         Dot Grid

Tile    1 ID:                    Loudoun_TM
Sampling Grid:                   f:/tour/sampling/loudoun_tm/loudoun_tm_grid.shp
Stratum File:                    f:/tour/sampling/loudoun_strata.img
Number of Strata:                61
Number of Samples:               15
Number of Samples Used:          15

Results

Area Estimate:                   52350.5 ha

Confidence Interval
Resampling Method:               Bootstrap
Number of Iterations:            10000

Median Value:                    51674.8 ha

                  Lower Bound    Upper Bound        CV        CV
90% Interval       48302.5 ha     55042.5 ha     -7.7%      5.1%
95% Interval       47655.5 ha     55651.9 ha     -9.0%      6.3%

Overall Estimate By Stratum
Stratum                 Percent          Statio-        Stratum        Percent           MOI
ID                      Sampled          narity            Area            MOI          Area
Stratum    1              2.08%            12.3         34.7 ha        100.00%       34.7 ha
Stratum    2              0.00%             0.0         52.4 ha          0.00%        0.0 ha
Stratum    3              0.34%             0.0        106.0 ha          0.00%        0.0 ha
Stratum    4              0.00%             0.0         15.6 ha          0.00%        0.0 ha
Stratum    5              0.61%            18.0        235.9 ha        100.00%      235.9 ha
Stratum    6              0.00%             0.0         80.3 ha          0.00%        0.0 ha
Stratum    7              0.37%            36.0        293.7 ha        100.00%      293.7 ha
Stratum    8              0.63%            15.8         57.1 ha        100.00%       57.1 ha
Stratum    9              0.40%            23.2        538.2 ha         50.00%      269.1 ha
Stratum   10              0.48%            25.6        149.2 ha        100.00%      149.2 ha
```

The Final Analysis Report gives a wealth of information about the Sampling project up to this point. It can indicate which of the strata are undersampled and which of the strata lack stationarity. Both of these issues must be addressed to achieve an accurate estimation of the land covered by the MOI.

Two Analysis Problems: Stationarity and Undersampling

The first few iterations of any Frame Sampling project serve mainly to reveal where the project breaks down. The Final Analysis Report reveals two of the biggest stumbling blocks for any project: stationarity and undersampling.

Stationarity

Stationarity, or Spatial Stationarity, is the measure of the MOI consistency in each stratum. A low Stationarity value means that the stratum reported relatively consistent MOI content percentages during the resampling iterations.

Undersampled strata

Not every stratum will be include areas that are sampled with high-resolution imagery, and some of those strata that are included will only have a very small percentage of the actual area that is sampled--not enough to make an accurate estimate of the MOI. These areas are said to be Undersampled.

Resolving the Problems

There are a number of ways to reduce the Undersampled strata and the Stationarity of the Strata; two of the most helpful methods are described below:

- Use the **Dendrogram Tool** (in the **Class Grouping Tools**) to revise your stratum file and group some of the problematic strata into spectrally similar groups that are adequately sampled. You will also need to recode the stratum file again.
- Some of the Strata may include classes that are substantially different from each other. These classes need to be split apart into two separate strata. Use the **Dendrogram Tool** (in the **Class Grouping Tools**) to revise your stratum file and regroup these classes.
- Add additional Samples to the project to get a better estimation of the contents of the strata.

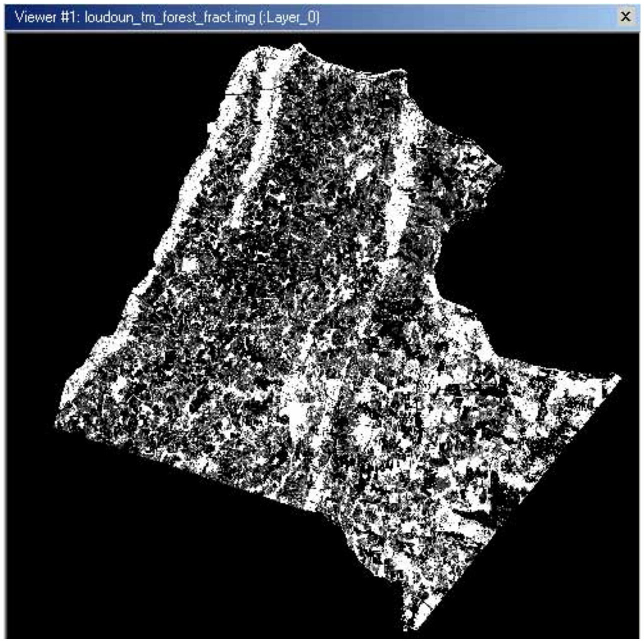
Resolving these problems to achieve an acceptable Confidence Value may require numerous iterations of refining and recoding the Stratum File, adding and/or removing Samples, as well as finding and correcting labelling errors in the Interpretation files.

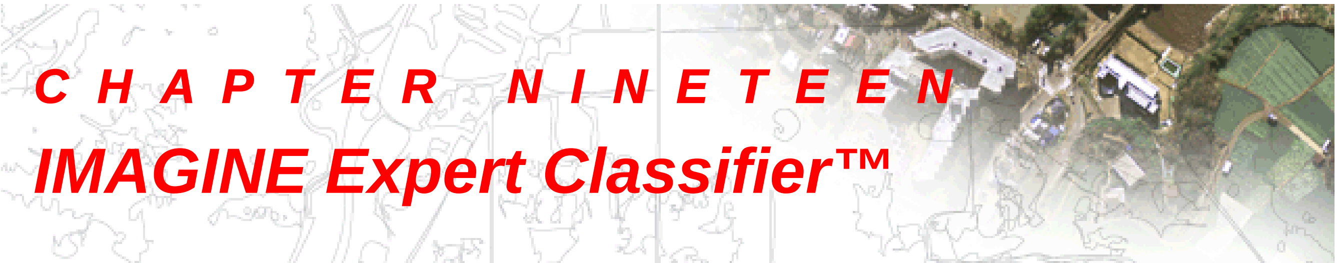
14. Once you are satisfied with the Analysis Results, click **Next** on the **Final Analysis Wizard** to generate the Fraction File.

The Final Analysis process will generate a Fraction File for each of the image tiles.

15. Click **Close** on the **Final Analysis Wizard** to exit the wizard and return to **Project Manager**.

The Fraction File, which is generated during the Final Analysis process, is a floating point file; each pixel value represents the probability of that pixel containing the MOI.


Viewer #1: loudoun_tm_forest_fract.img (:Layer_0)



CHAPTER NINETEEN

IMAGINE Expert Classifier™

Introduction

This chapter is designed to introduce you to the IMAGINE Expert Classifier™. The IMAGINE Expert Classifier is composed of two modules: the Knowledge Engineer and the Knowledge Classifier. The Knowledge Engineer provides the interface for an expert with first-hand knowledge of the data and the application to identify the variables, rules, and output classes of interest and create the hierarchical decision tree. The Knowledge Classifier provides an interface for a non-expert to apply the knowledge base and create the output classification.

This set of exercises guides you through the basic process of creating a new knowledge base from scratch. The Knowledge Engineer tools and their uses are presented.

Create a Knowledge Base

In this tour guide you can learn how to:

- add hypotheses
- enter rules for hypotheses
- edit variables for the rules
- copy and edit existing rules
- test a knowledge base

Approximate completion time for this tour guide is 30 minutes.

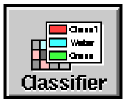
Set Up the Output Classes

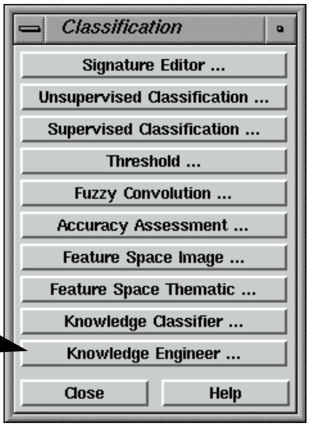
For the purpose of this exercise, suppose that you are determining Residential and Commercial Services map classes from imagery and existing mapped data. (The example classes are a subset of the **lanier.ckb** provided in the examples directory.)

This very simple two class example provides an opportunity to use and become familiar with the tools and processes of the Knowledge Engineer. The Knowledge Engineer aids in the process of designing a knowledge base by allowing you to set up a framework which can be easily be edited and rearranged during the design process.

Start the Knowledge Engineer

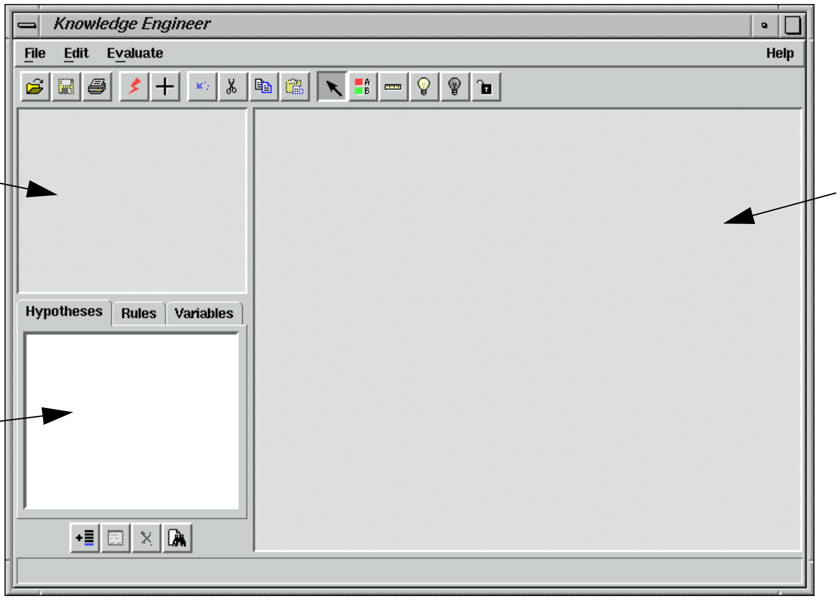
1. Click the Classifier icon and select **Knowledge Engineer** from the **Classification** menu.


Class 1
Water
Grass
Classifier


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Close
Help

Click here to start the Knowledge Engineer

The Knowledge Engineer dialog starts with blank slates in the edit window, the decision tree overview section, and the Knowledge Base component list (**Hypotheses**, **Rules**, and **Variables**).


Knowledge Engineer
File
Edit
Evaluate
Help
Hypotheses
Rules
Variables

Decision tree overview section

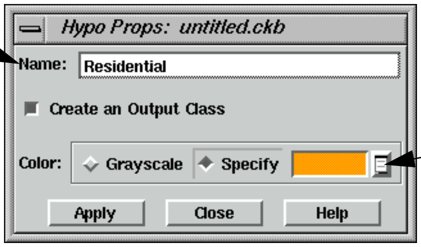
Edit window

Knowledge Base component list

Place Hypotheses into the Edit Window

1. Select **Edit | New Hypothesis** to add the first hypothesis.

The Hypo Props (Hypothesis Properties) dialog opens with **untitled.ckb** in the title bar, a default hypothesis name: **New Hypothesis**, and the **Color** is set to **Grayscale**.



Type the new class name here

Specify the color Orange

2. Change the default hypothesis **Name** to the first class name, **Residential**.
3. Since you want **Residential** to be an output class, the **Create an Output Class** checkbox is left checked.

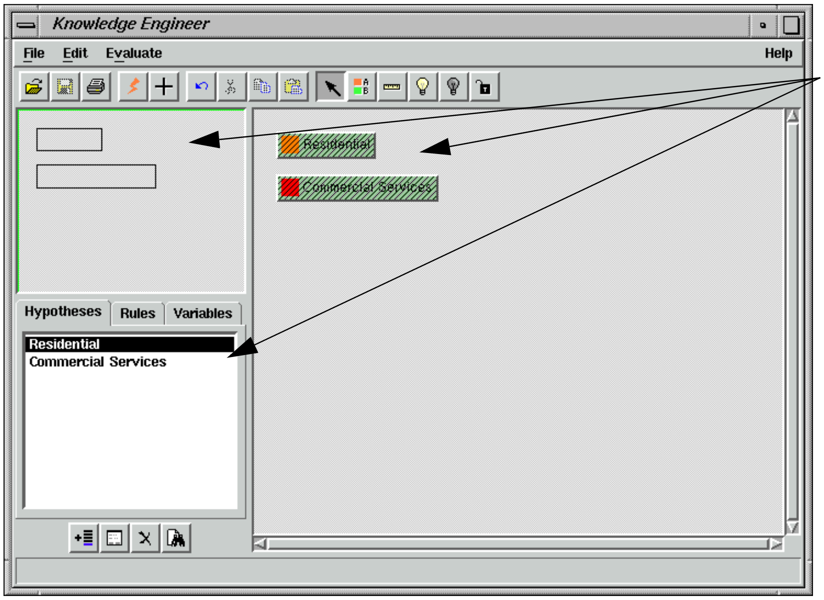
   You are going to give colors to each of the classes.
4. Click the **Specify** radio button in the **Color** section. Then use the pulldown menu to select **Orange** as the color for this class.

Selecting Colors for Output Classes

If a color is not specified for an output class, it is automatically made grayscale. As additional grayscale output classes are added, grayscale values for each of the grayscale classes are automatically updated and stretched evenly across the range from white to black. This occurs even if some other classes are assigned specific colors.

5. Now click the **Apply** button in the Hypo Props dialog.

   A green rectangle with the hypothesis name **Residential** and chip color appears in the edit window and an outline of the rectangle appears in the knowledge tree overview window. You probably noticed that there are diagonal lines through the hypothesis rectangle in the edit window. These lines remain until conditions have been added that can make the hypothesis true or false.
6. Select **Edit | New Hypothesis** once again to set up the next class, **Commercial Services**. Enter the class **Name** and **Specify Red** as the color for the class.
7. Click **Apply** in the Hypo Props dialog to add the class.
8. Click **Close** on the Hypo Props dialog.


Knowledge Engineer
File
Edit
Evaluate
Help
Residential
Commercial Services
Hypotheses
Rules
Variables
Residential
Commercial Services

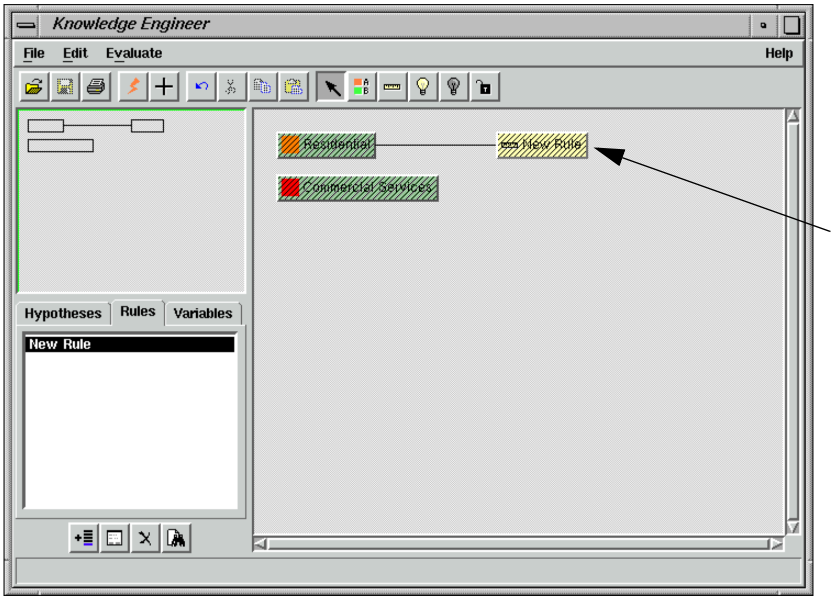
New classes display in the edit window, in the overview, and in the component list

Enter Rules for the Hypothesis

1. Select the Create Rule Graphic Tool icon from the Knowledge Engineer dialog icon bar.

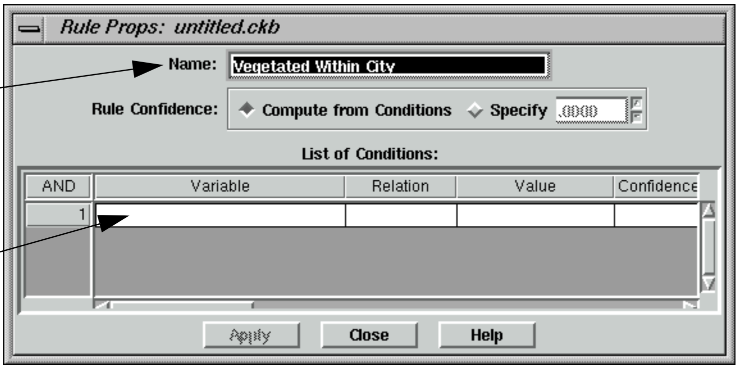
2. Move the cursor, which changes to the shape of a rule, and click on the green hypothesis rectangle for **Residential**.

   A yellow rule rectangle, called **New Rule**, is attached to the hypothesis rectangle, **Residential**, by a line that is mirrored in the knowledge tree overview.


Knowledge Engineer
File
Edit
Evaluate
Help
Residential
New Rule
Commercial Services
Hypotheses
Rules
Variables
New Rule

A new rule is added to the Residential class

3. Double-click the yellow **New Rule** rectangle to open the Rule Props (Rule Properties) dialog.


Rule Props: untitled.ckb
Name:
Rule Confidence:
Compute from Conditions
Specify
List of Conditions:
AND
Variable
Relation
Value
Confidence
1
Apply
Close
Help

Name: Vegetated Within City

Rule Confidence: ● Compute from Conditions ○ Specify 0.000

Enter the name for the new rule here

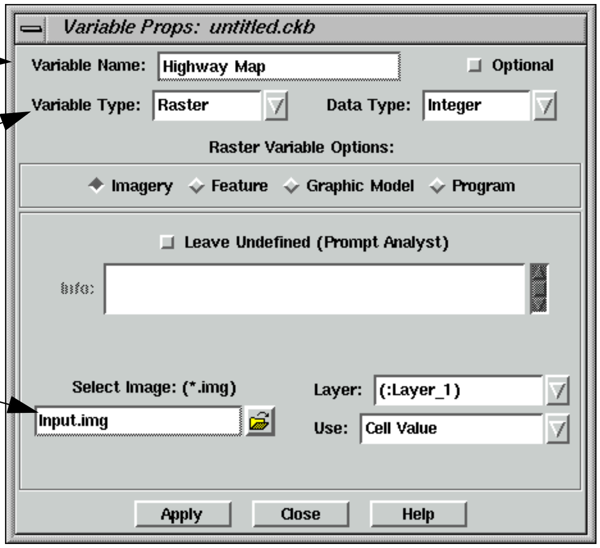
Click here to create a new variable

4. Change the **Name** of the rule to **Vegetated Within City** and leave the **Compute from Conditions** radio button selected for **Rule Confidence**.

Enter Variables for the Rule

1. Click within the cell under **Variable** and select **New Variable** from the popup list.

   The Variable Props dialog opens.



Type the name of the variable here

Select Raster as the Variable Type

Select Input.img here

Variable Props: untitled.ckb
☐ Optional
Raster Variable Options:
● Imagery ○ Feature ○ Graphic Model ○ Program
☐ Leave Undefined (Prompt Analyst)
Info:
Apply
Close
Help

Variable Name: Highway Map

Variable Type: Raster

Data Type: Integer

Select Image: (*.img) Input.img

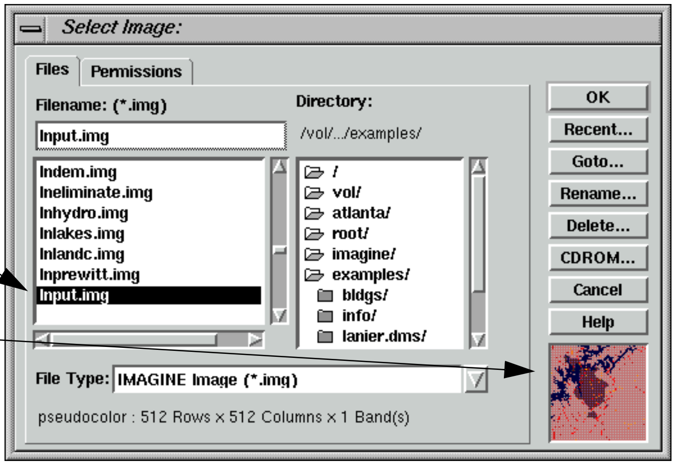
Layer: (:Layer_1)

Use: Cell Value

2. Change the **Variable Name** to **Highway Map**, and change the **Variable Type** to **Raster**.

   Changing the type to **Raster** switches the bottom part of the dialog to the **Raster Variable Options**, providing a different set of choices than for the **Scalar** variable type.

3. Click the Select Image File icon, then navigate to and select **Input.img** from the <IMAGINE_HOME>/examples directory.



Select Input.img

A preview displays

Select Image:
Files
Permissions
Indem.img
Ineliminate.img
Inhydro.img
Inlakes.img
Inlandc.img
Inprewitt.img
Input.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) Input.img

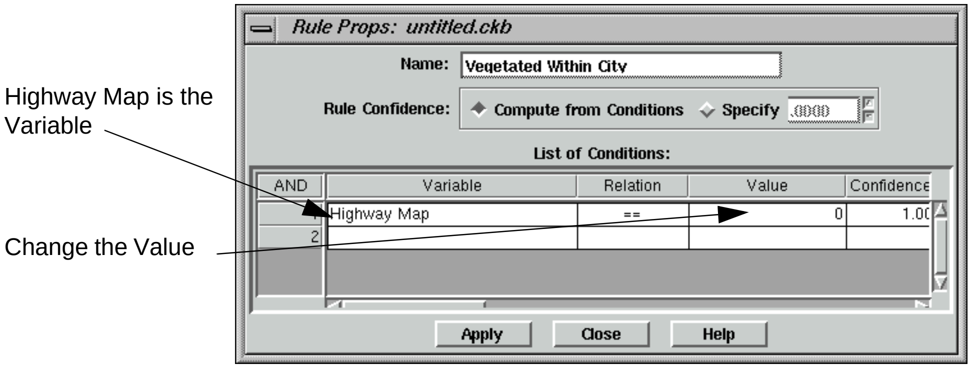
Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

pseudocolor : 512 Rows x 512 Columns x 1 Band(s)

4. Click **OK** in the Select Image dialog to add the file to the Variable Props dialog.

5. Click the **Apply** button in the Variable Properties dialog to add **Highway Map** to the rule properties CellArray.



Name: Vegetated Within City

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Variable	Relation	Value	Confidence
1	Highway Map	==	0	1.00
2				

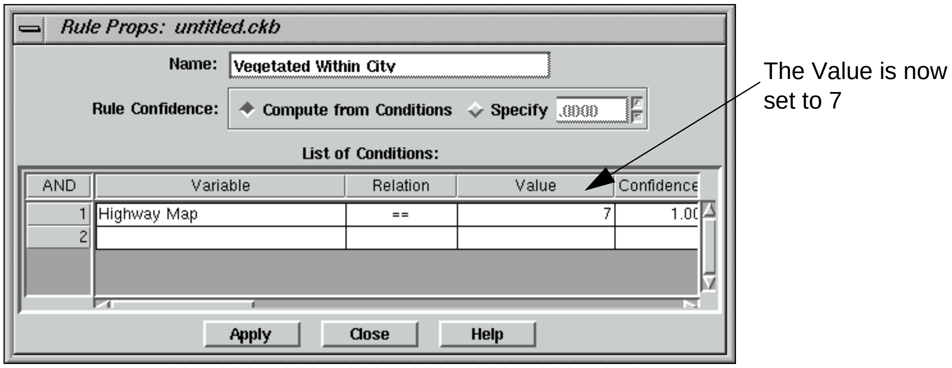
Highway Map is the Variable

Change the Value

6. Click **Close** to dismiss the Variable Props dialog.

7. In the Rule Props dialog, click in the cell under **Value** and select **Other**.

8. Into the highlighted cell, type **7** and press Return on your keyboard (7 is the class number for urban areas in **Input.img**).



Name: Vegetated Within City

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

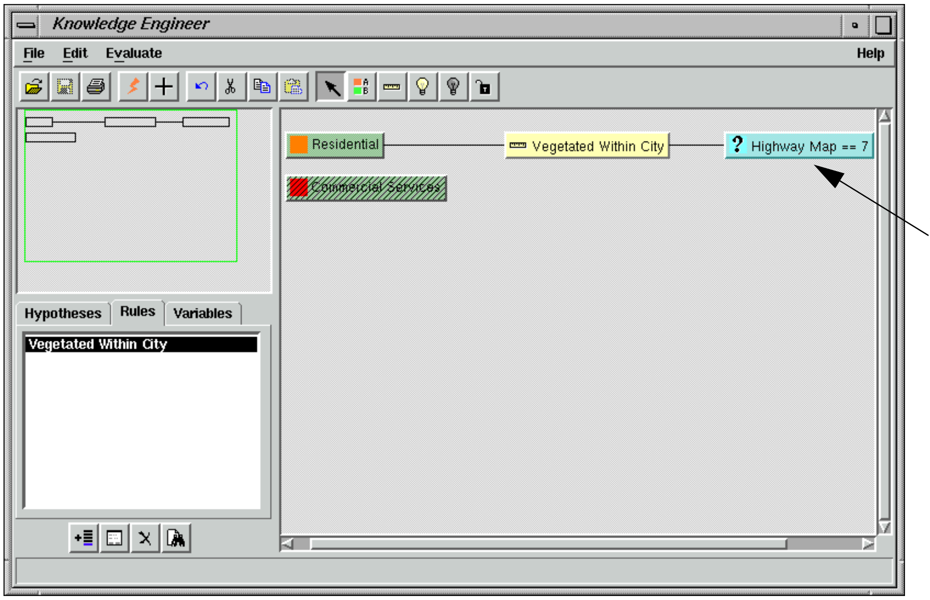
List of Conditions:

AND	Variable	Relation	Value	Confidence
1	Highway Map	==	7	1.00
2				

The Value is now set to 7

9. Click **Apply** in the Rule Props dialog to enter the changes, then **Close**.

   The new rule with its attached variable appears in the edit window. Notice that the diagonal lines in the hypothesis, **Residential**, and rule, **Vegetated Within City**, rectangles have disappeared for the hypothesis and rule you have edited. This is because at least one complete condition is now set.


Knowledge Engineer
File
Edit
Evaluate
Help
Residential
Vegetated Within City
? Highway Map == 7
Commercial Services
Hypotheses
Rules
Variables
Vegetated Within City

The Variable Highway Map has been added to the tree

Add an Intermediate Hypothesis

In this section, you add an intermediate hypothesis as well as its conditions.

1. Select the Create Hypothesis graphic tool and click on the rule, **Vegetated Within City**.

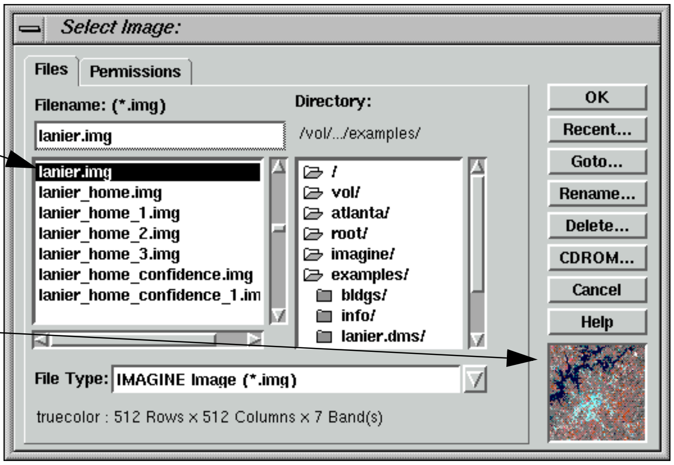
   An intermediate hypothesis, **New Hypothesis**, is attached to the rule, linked by a **New Hypothesis == TRUE** variable.

2. Double-click the **New Hypothesis** rectangle to open the Hypo Props dialog.
3. In the Hypo Props dialog, change the name to **Vegetation** and deselect the **Create an Output Class** checkbox since you do not want this to be an output class.
4. Click **Apply**, then **Close**.

Create a New Rule

1. Using the Create Rule graphic tool, place a **New Rule** on the **Vegetation** hypothesis.
2. Double-click on the **New Rule** to open the Rule Props dialog, and change the rule **Name** to **High IR and Low Visible**.

3. Click in the cell below **Variable** and select **New Variable**.
4. Type the name **TM Band 4** in the **Variable Name** field.
5. Change the **Variable Type** to **Raster**
6. Click the Open icon to open the Select Image dialog, and select **lanier.img** from the <IMAGINE_HOME>/examples directory.


Select Image:
Files
Permissions
lanier.img
lanier_home.img
lanier_home_1.img
lanier_home_2.img
lanier_home_3.img
lanier_home_confidence.img
lanier_home_confidence_1.im
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.img) lanier.img

Directory: /vol/.../examples/

File Type: IMAGINE Image (*.img)

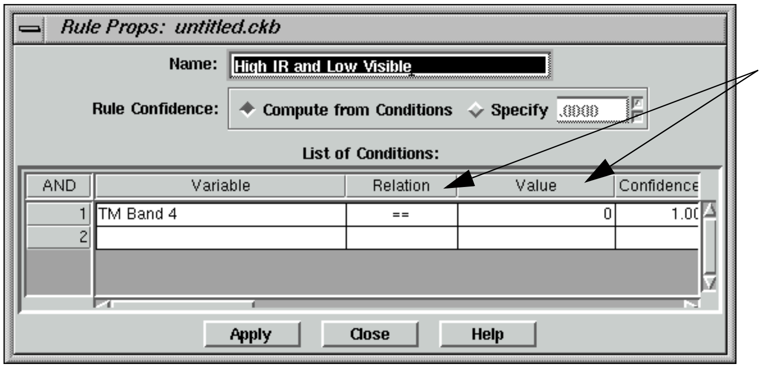
truecolor : 512 Rows x 512 Columns x 7 Band(s)

Select lanier.img

A preview displays

7. Click **OK** in the Select Image dialog to add **lanier.img** to the Variable Props dialog.
8. Click the **Layer** dropdown list and select **(:Layer_4)**.
9. Click **Apply,** then **Close** in the Variable Props dialog.

   The Rule Props dialog updates.


Rule Props: untitled.ckb
Apply
Close
Help

Name: High IR and Low Visible

Rule Confidence: ◆ Compute from Conditions ◇ Specify .8000

List of Conditions:

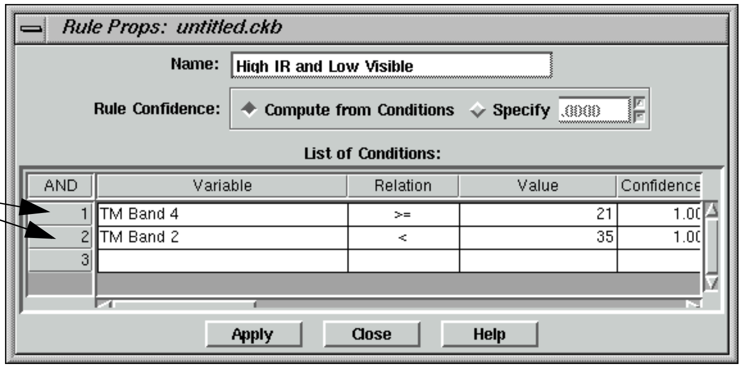
AND	Variable	Relation	Value	Confidence
1	TM Band 4	==	0	1.0
2				

Now that you have created the Variable, change its Relation and Value

10. In the Rule Props dialog, click in the cell below **Relation** and select **>=**.
11. Click, then select **Other** from the **Value** cell, change the **Value** to **21**, then press Return on your keyboard.

12. Now, using step 3. through step 11. above, add layer 2 of **lanier.img** as the second variable (row **2** under the **AND** column), name it **TM Band 2**, set **Relation** to **<** and set the value to **35**.

Two Variables, their Relations and Values have been added to the rule High IR and Low Visible



Rule Props: untitled.ckb

Name: High IR and Low Visible

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Variable	Relation	Value	Confidence
1	TM Band 4	>=	21	1.0
2	TM Band 2	<	35	1.0
3				

13. Click **Apply**, then **Close** in the Rule Props dialog.

Copy and Edit

Since the hypothesis for the **Commercial Services** class has very similar rules and conditions to the **Residential** class, some of the conditions can be used directly, or copied and edited to save time.

1. Begin editing the **Commercial Services** class by placing a new rule on the **Commercial Services** hypothesis rectangle, then double-clicking the **New Rule** to open the Rule Props dialog.

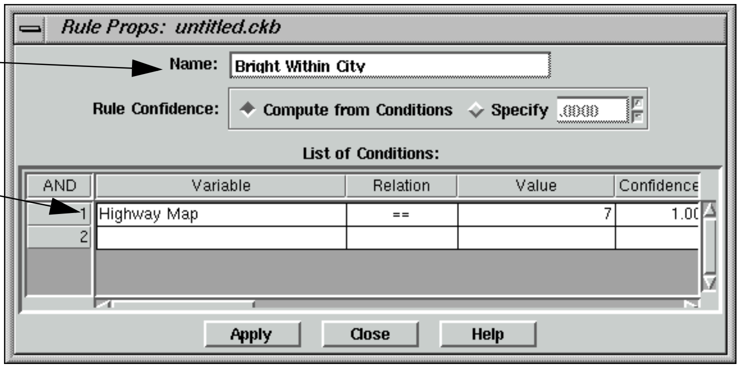
Refer to "Enter Rules for the Hypothesis"*on page 562 if you forget how to create a new rule.*

2. In the Rule Props dialog, change the **Name** of the rule to **Bright Within City**.

The first variable that is needed is **Highway Map**, which is now in the **Variable** list since it was entered previously.

3. Click in the cell below **Variable** and select **Highway Map**, confirm that the **Relation** is set to **==**, and set the **Value** to **7**.

As before, this makes the variable equal to the urban area from **Input.img**.


Rule Props: untitled.ckb
List of Conditions:
Apply
Close
Help

Name: Bright Within City

Rule Confidence: ● Compute from Conditions ○ Specify .0000

AND	Variable	Relation	Value	Confidence
1	Highway Map	==	7	1.00
2				

The Commercial Services class has the rule Bright Within City, which has the Variable Highway Map.

4. Click **Apply** in the Rule Props dialog, then **Close**.

5. Now use the Create Hypothesis graphic tool to place a new hypothesis (which is an intermediate hypothesis) on the **Bright Within City** rule rectangle.

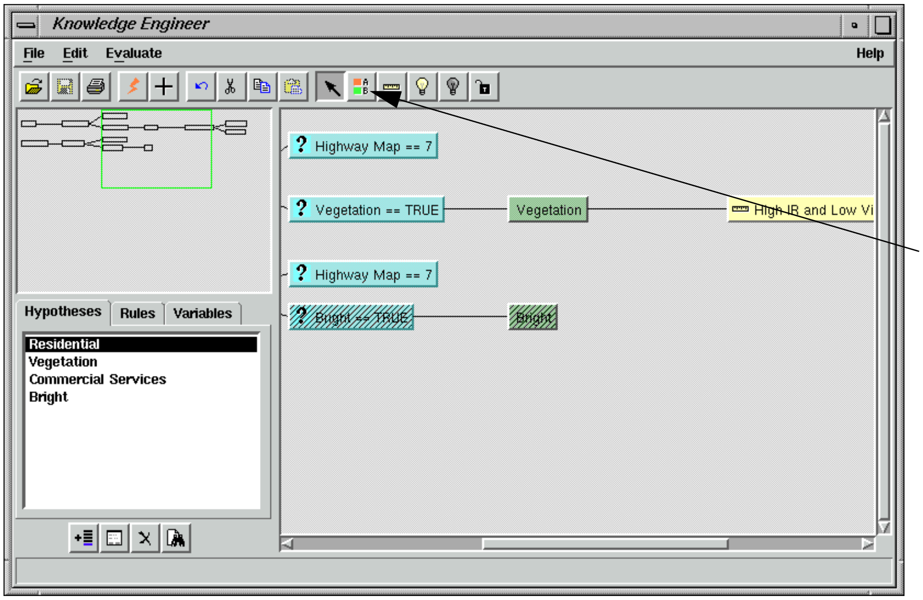
See "Add an Intermediate Hypothesis"*on page 566 if you forgot how to create a hypothesis.*

6. Double-click on the **New Hypothesis** to open the Hypo Props dialog.

7. In the Hypo Props dialog, name the new hypothesis **Bright** and deselect the **Create an Output Class** checkbox.

8. Click **Apply**, then **Close** in the Hypo Props dialog.

   The Knowledge Engineer dialog updates accordingly.


Knowledge Engineer
File
Edit
Evaluate
Help
Highway Map == 7
Vegetation == TRUE
Vegetation
High IR and Low Vi
Highway Map == 7
Bright == TRUE
Bright
Hypotheses
Rules
Variables
Residential
Vegetation
Commercial Services
Bright

The Bright Hypothesis has been added using the Create Hypothesis icon

Since the rule to be attached to the **Bright** hypothesis is very similar to the **High IR and Low Visible** rule that is attached to the **Vegetation** hypothesis, you can make a copy of it to paste and edit.

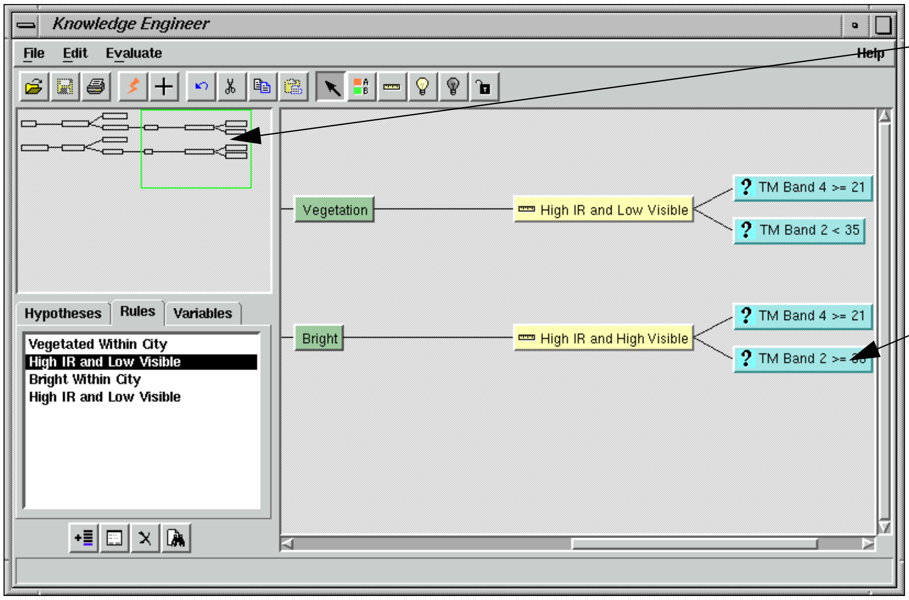
9. Click on the **High IR and Low Visible** rule.
10. Right-click, and select **Copy** from the **Options** menu.
11. Click on the **Bright** hypothesis, then right-click and select **Paste** from the **Options** menu.

    A new rule is attached to the **Bright** hypothesis with a default name of **High IR and Low Visible (1)** (the **(1)** is added since it is a copy).

12. Double-click the **High IR and Low Visible (1)** rule to open the Rule Props dialog.
13. In Rule Props dialog for the new rule, change the **Name** to **High IR and High Visible**.

    The only change that needs to be made to the variables is the **Relation** for **TM Band 2**.

14. Change the **Relation** for **TM Band 2** to **>=**.
15. Click **Apply**, then **Close** in the Rule Props dialog.


Knowledge Engineer
File
Edit
Evaluate
Help
Hypotheses
Rules
Variables
Vegetated Within City
High IR and Low Visible
Bright Within City
High IR and Low Visible
Vegetation
High IR and Low Visible
TM Band 4 >= 21
TM Band 2 < 35
Bright
High IR and High Visible
TM Band 4 >= 21
TM Band 2 >= 35

The portion of the tree visible in the edit window is highlighted in the overview window

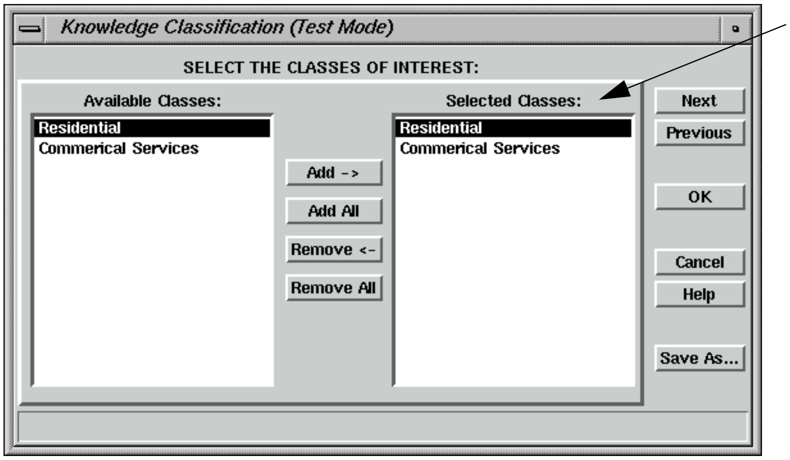
Rule properties have been changed for the High IR and High Visible rule

At this point, two hypotheses and their conditions have been entered. Now, the two classes can be tested to see what pixels are allocated to them.

Test the Knowledge Base

1. On the Knowledge Engineer dialog tool bar, select the Run Test Classification icon (or select **Evaluate | Test Knowledge Base**).

The Knowledge Classification dialog opens in Test Mode at the **SELECT THE CLASSES OF INTEREST** panel, along with a new Viewer where the test classification is displayed. All active enabled classes are selected by default.


Knowledge Classification (Test Mode)
SELECT THE CLASSES OF INTEREST:
Available Classes:
Residential
Commerical Services
Add ->
Add All
Remove <-
Remove All
Selected Classes:
Residential
Commerical Services
Next
Previous
OK
Cancel
Help
Save As...

The Selected Classes are Residential and Commercial Services

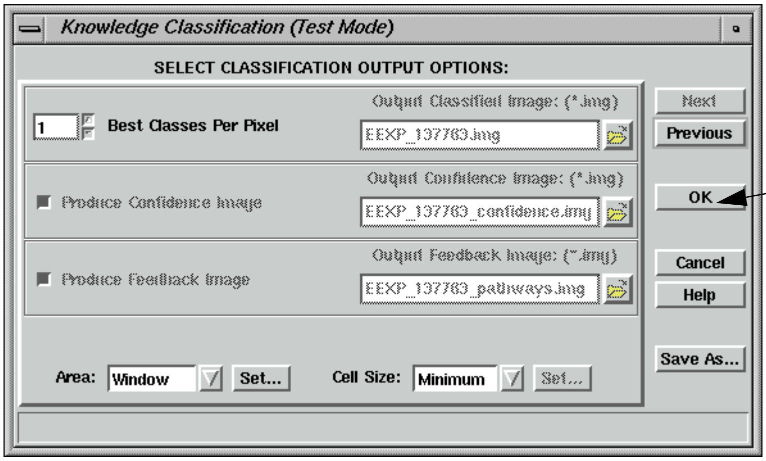
2. Leave the two classes, **Residential** and **Commercial Services**, selected in the **Selected Classes** section of the Knowledge Classification dialog.

3. Click **Next** to go to the next panel of the Knowledge Classification dialog.

If the **Prompt User** *option had been selected instead of entering file names for the variables, an intermediate panel, SELECT THE INPUT DATA FOR CLASSIFICATION, would appear here to allow entry of file names.*

The Select Classification Output Options panel allows you to set the number of best classes per pixel, set an output area and set an output cell size. The defaults are used here since you only have two classes and small images that are the same size and have the same cell size.

Also note the grayed-out options for Output Classified Image, Output Confidence Image, and Output Feedback Image. These images are made temporary files in Test Mode, but can be selected as output files when running Knowledge Classifier in regular (non-test) mode from the* Classification *menu.


Knowledge Classification (Test Mode)
SELECT CLASSIFICATION OUTPUT OPTIONS:
Next
Previous
OK
Cancel
Help
Save As...
Set...
Set...

Best Classes Per Pixel: 1

Output Classified Image: (*.img) EEXP_137763.img

☐ Produce Confidence Image — Output Confidence Image: (*.img) EEXP_137763_confidence.img

☐ Produce Feedback Image — Output Feedback Image: (*.img) EEXP_137763_pathways.img

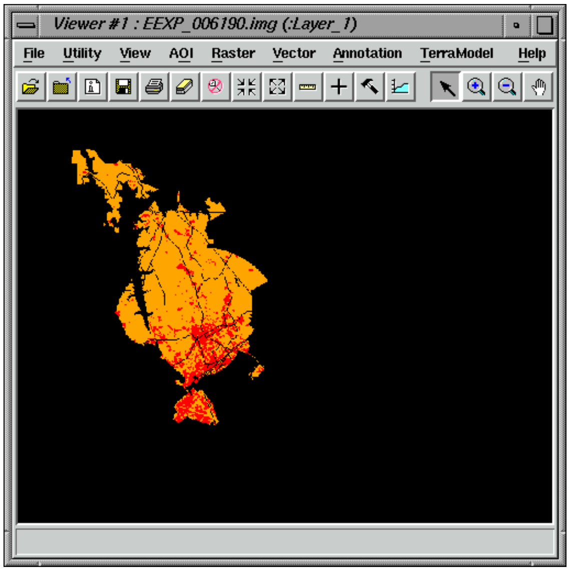
Area: Window

Cell Size: Minimum

Click OK to generate the test classification

4. Click **OK** in the Knowledge Classification dialog to start the test classification.

   A status bar opens. When the classification has completed, the test classification image is displayed in the Viewer.

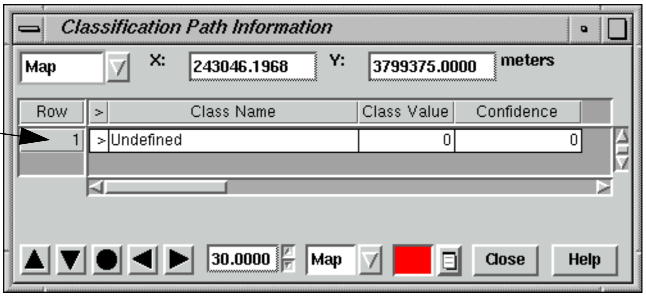

Viewer #1 : EEXP_006190.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

5. Click **OK** to dismiss the status bar when the classification is finished.

6. In the Knowledge Engineer dialog, click on the Start Classification Pathway Feedback Mode icon.

   +

   The Classification Path Information dialog is opened along with a cursor in the Viewer.


Classification Path Information
Close
Help

Map X: 243046.1968 Y: 3799375.0000 meters

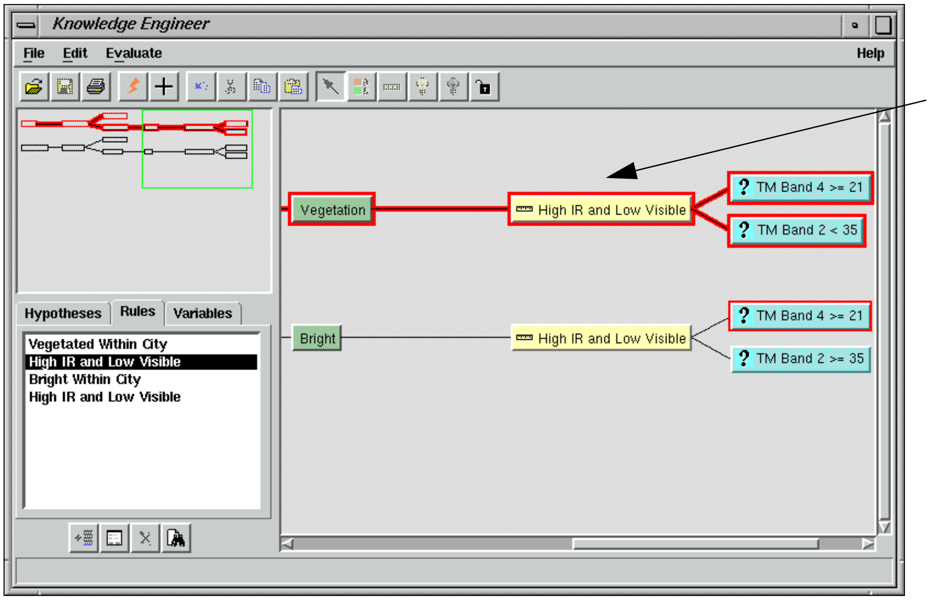
Row	>	Class Name	Class Value	Confidence
1	>	Undefined	0	0

30.0000 Map

Details about the class under the cursor display here

7. Move the cursor into the orange and red areas in the Viewer, which correspond to the orange **Residential** class and the red **Commercial Services** class.

   Note that when the cursor is placed on a pixel for one of the classes, the path for the class is highlighted in the Knowledge Engineer dialog and in the overview window. In complex knowledge bases, this feature is useful for telling at a glance which hypothesis was used to classify the point of interest.

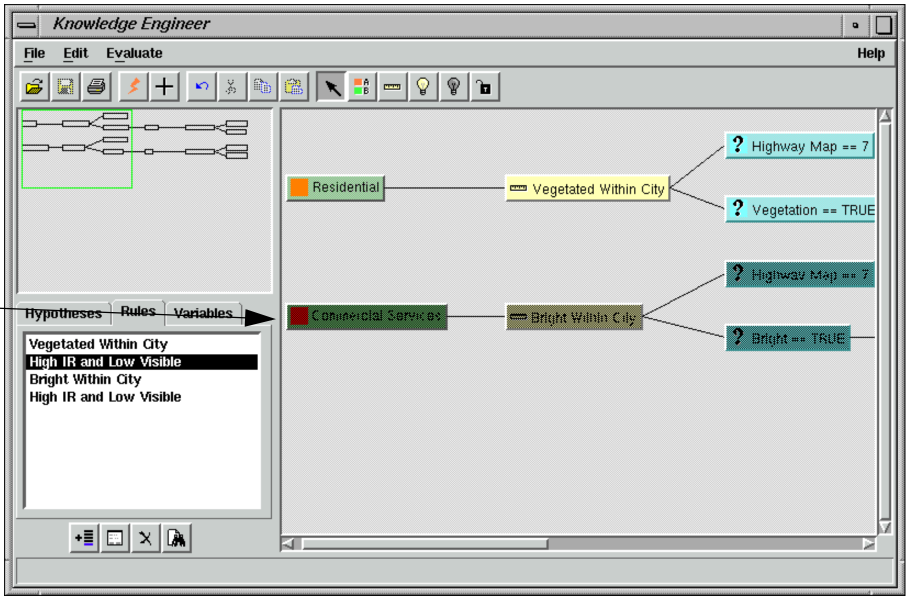

Knowledge Engineer
File
Edit
Evaluate
Help
Hypotheses
Rules
Variables
Vegetated Within City
High IR and Low Visible
Bright Within City
High IR and Low Visible
Vegetation
High IR and Low Visible
TM Band 4 >= 21
TM Band 2 < 35
Bright
High IR and Low Visible
TM Band 4 >= 21
TM Band 2 >= 35

The class currently under the cursor is highlighted in the Knowledge Engineer dialog

8. Click the **Close** button to dismiss the Classification Path Information dialog.

9. Select the gray Disable Node icon, then click on the **Commercial Services** hypothesis graphic to disable it.

The Commercial hypothesis path is grayed-out. This means the class is not considered when a test classification is run (or, in the regular Knowledge Classifier, if the knowledge base has been saved with the class disabled).


Knowledge Engineer
File
Edit
Evaluate
Help
Hypotheses
Rules
Variables
Residential
Vegetated Within City
Highway Map == 7
Vegetation == TRUE
Commercial Services
Bright Within City
Highway Map == 7
Bright == TRUE

Vegetated Within City
High IR and Low Visible
Bright Within City
High IR and Low Visible

Commercial Services is disabled

10. To enable the **Commercial Services** class once again, click on the **Commercial Services** hypothesis graphic with the yellow Enable Node icon (or right-click on the hypothesis graphic and select **Enable**).

11. Save the knowledge base by selecting **File | Save As.**

12. Navigate to a directory in which you have write permission, and name the file **ResComm_Class.ckb**.

13. Click **OK** in the Save Classification Knowledge Base As dialog.

14. Select **File | Close** from the Knowledge Engineer dialog, which is now entitled **ResComm_Class.ckb**, to finish.

Create a Portable Knowledge Base

This exercise is going to give you practice creating and using a portable knowledge base. In this example, you use a knowledge base to determine areas most suitable for cross-country travel.

Data

Data available for the project includes the following:

- a landcover classification (**supervised.img**)
- a DEM (**30meter.img**)
- a map of minor and major roads (**roads.img**)
- a near-infrared degraded air photo with 30m resolution (**mason_ap.img**)

The file **supervised.img** shows a typical landcover classification derived from Landsat TM data (a portion of the Landsat scene is provided as **tm_860516.img** along with the signature file **tm_860516.sig**, which was used to produce a maximum likelihood classification). The image shows the distribution of broad landcover categories such as different types of forestry, human-made features, water and open ground. However, it does not show the land use of each pixel, or how each pixel could be put to use.

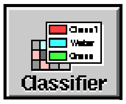
Consider a scenario whereby someone wishes to traverse this area of ground with one or more vehicles. They need to use the landcover information, along with other ancillary data to help determine which areas can be traversed easily and which cannot.

Methodology

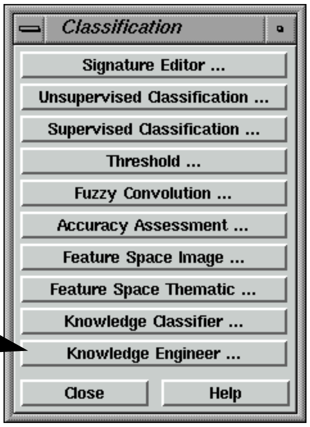
Given these data sets, we can start to envisage expert rules that are based on these data (and data derived from them) to determine the ease of crossing a particular area.

ERDAS IMAGINE must be running.

1. Click the Classifier icon in the ERDAS IMAGINE icon panel.


Class1
Water
Grass
Classifier

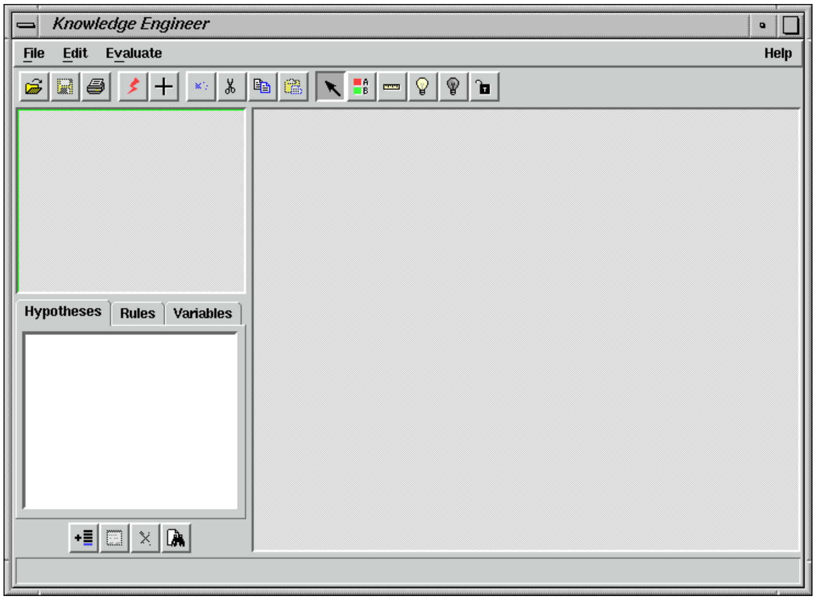
The **Classification** menu opens.


Classification
Signature Editor ...
Unsupervised Classification ...
Supervised Classification ...
Threshold ...
Fuzzy Convolution ...
Accuracy Assessment ...
Feature Space Image ...
Feature Space Thematic ...
Knowledge Classifier ...
Knowledge Engineer ...
Close
Help

Click here to start the Knowledge Engineer

2. Click **Knowledge Engineer** in the **Classification** menu.

   An empty Knowledge Engineer dialog opens.



Open a Knowledge Base

Next, you can open the **mobility_factors.ckb** knowledge base to examine what expert rules are used and how their components were created.

1. In the Knowledge Engineer dialog, click the Open icon, or select **File | Open**.

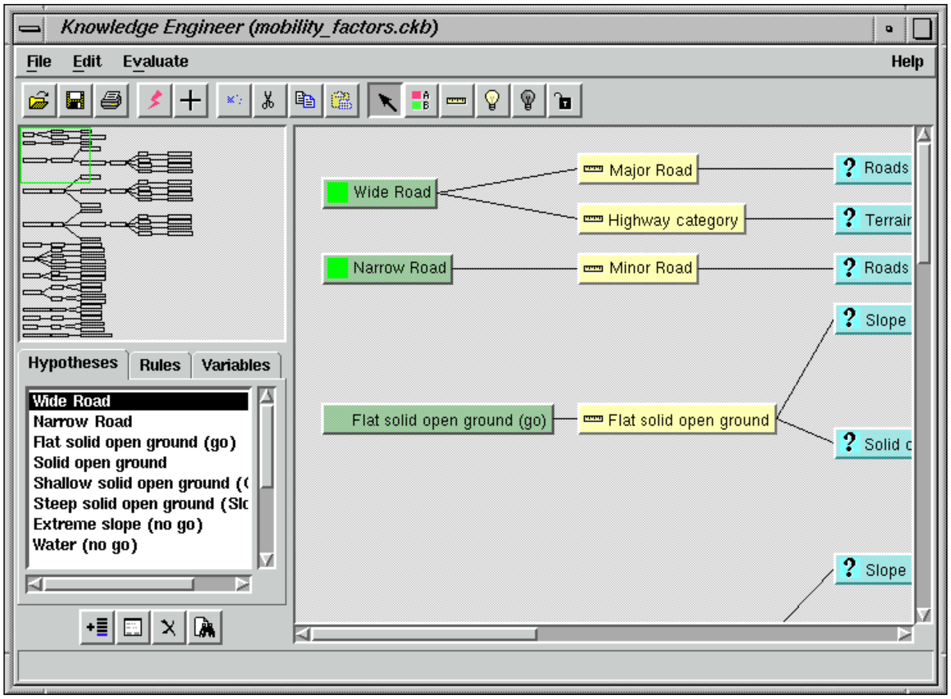
   The Open Classification Knowledge Base dialog opens.

2. Navigate to the <IMAGINE_HOME>/examples directory, and select the file **mobility_factors.ckb**.

   <IMAGINE_HOME> is the directory where ERDAS IMAGINE resides on your system.

3. Click **OK** in the Open Classification Knowledge Base dialog to load the file.

   The knowledge base of mobility factors opens.


Knowledge Engineer (mobility_factors.ckb)
File
Edit
Evaluate
Help
Wide Road
Major Road
Roads
Highway category
Terrain
Narrow Road
Minor Road
Roads
Slope
Flat solid open ground (go)
Flat solid open ground
Solid c
Slope
Hypotheses
Rules
Variables
Wide Road
Narrow Road
Flat solid open ground (go)
Solid open ground
Shallow solid open ground (
Steep solid open ground (Sl
Extreme slope (no go)
Water (no go)

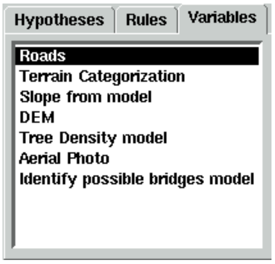
Examine the Knowledge Base

This knowledge base was created by defining as many of the variables as would be needed as possible. For example, roads are going to be the easiest areas to traverse, so a variable was needed to define where roads are that can be used.

1. In the Knowledge Engineer dialog, click on the **Variables** tab.

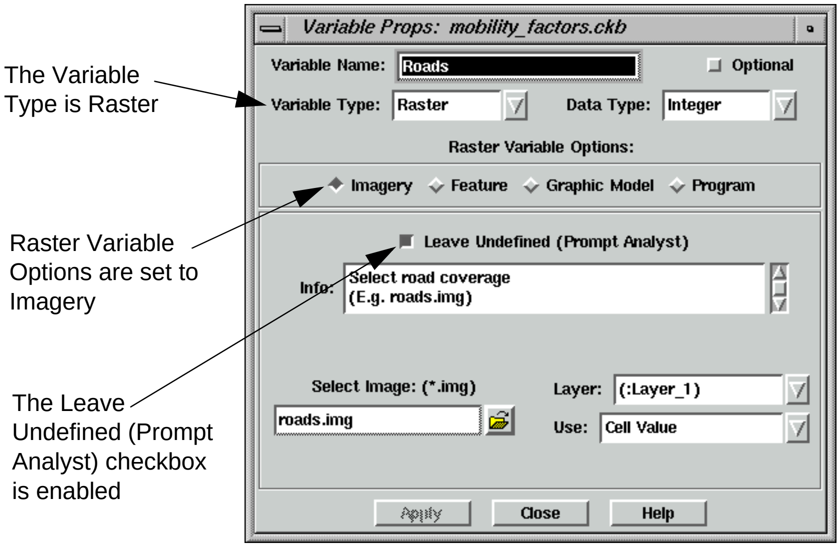
   The variables for the **mobility_factors.ckb** knowledge base display.

Double-click the Roads variable to see its properties


Hypotheses
Rules
Variables
Roads
Terrain Categorization
Slope from model
DEM
Tree Density model
Aerial Photo
Identify possible bridges model

2. In the **Variables** list, double-click the **Roads** variable.

   The Variable Props dialog opens.



Variable Props: mobility_factors.ckb

Variable Name: Roads

☐ Optional

Variable Type: Raster

Data Type: Integer

Raster Variable Options:

● Imagery ○ Feature ○ Graphic Model ○ Program

☑ Leave Undefined (Prompt Analyst)

Info: Select road coverage (E.g. roads.img)

Select Image: (*.img) roads.img

Layer: (:Layer_1)

Use: Cell Value

The Variable Type is Raster

Raster Variable Options are set to Imagery

The Leave Undefined (Prompt Analyst) checkbox is enabled

In the Variable Properties dialog, you can see that the **Variable Type** is **Raster** and the **Imagery** option is selected because the input is an image.

This knowledge base is transportable—you may want to pass it to a colleague in another office, or reuse it yourself to automate a production process. So, rather than selecting a specific image to be used, the **Leave Undefined** checkbox is selected and a prompt for the type of data that you want the end user to supply is typed in the **Info** window (i.e., **Select road coverage**).

The same type of imagery variables have been defined for the landcover classification (**Terrain Categorization**), Digital Elevation Model (**DEM**) and air photo (**Aerial Photo**).

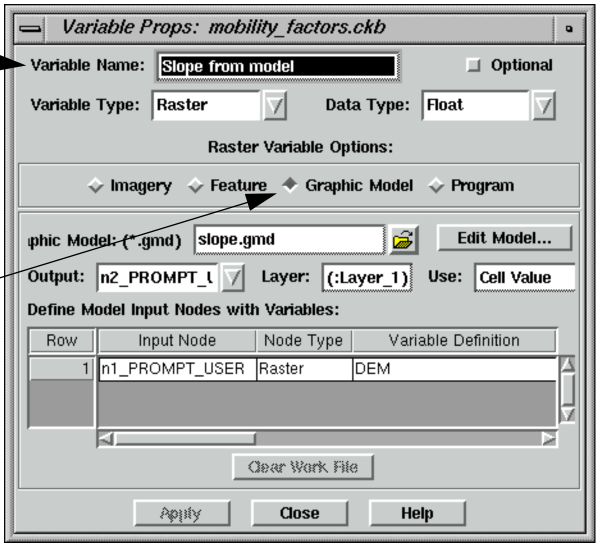
Some of the imagery variables are used directly in rules (such as the **Terrain Categorization** variable being used to identify open ground in the image). Others are used indirectly to calculate variable values.

For example, open ground (e.g., grass, scrub) is also good for vehicles to cross, but not if the ground is steeply sloping. The fact that an area is open ground can be determined from the landcover classification (the **Terrain Categorization** variable), but you do not have an image that provides slope directly. However, you do have a digital elevation model (the **DEM** variable), which can be used to derive slope.

Derive Slope Values

1. In the **Variables** tab, double-click the variable called **Slope from model**.

The Variable Props dialog updates accordingly.



These are the properties for the Slope from model variable

The Slope from model variable uses a Graphic Model

Variable Props: mobility_factors.ckb

Variable Name: Slope from model

☐ Optional

Variable Type: Raster

Data Type: Float

Raster Variable Options:

○ Imagery ○ Feature ● Graphic Model ○ Program

phic Model: (*.gmd) slope.gmd

Output: n2_PROMPT_

Layer: (:Layer_1)

Use: Cell Value

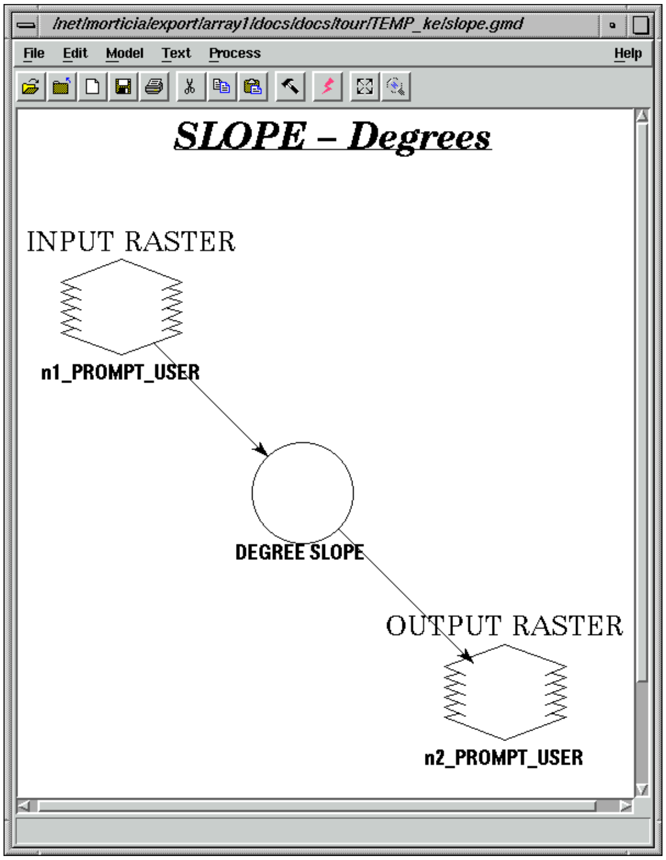
Define Model Input Nodes with Variables:

Row	Input Node	Node Type	Variable Definition
1	n1_PROMPT_USER	Raster	DEM

Notice that the variable is again raster in nature, so the **Variable Type** is set to **Raster**. In this case, however the **Graphic Model** option of the **Raster Variable Options** has been selected. The graphic model associated with this variable is named **slope.gmd**.

2. In the Variable Props dialog, click on the **Edit Model** button to view the graphic model.

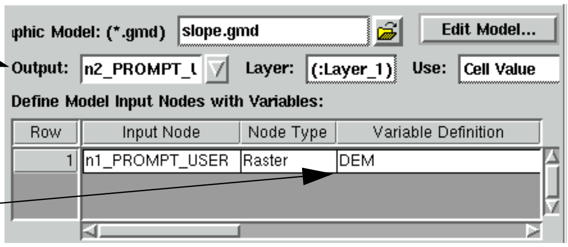
   A Spatial Modeler viewer opens, which contains the model that defines the **Slope from model** variable.


/net/morticia/export/array1/docs/docs/tour/TEMP_ke/slope.gmd
File
Edit
Model
Text
Process
Help
SLOPE – Degrees
INPUT RASTER
n1_PROMPT_USER
DEGREE SLOPE
OUTPUT RASTER
n2_PROMPT_USER

This is the model that the variable **Slope from model** uses to calculate the slope of any location.

To make the knowledge base transportable, you do not want to define actual image names in the **slope.gmd** model. Instead, the **INPUT RASTER** and the **OUTPUT RASTER** of the model have been set to **PROMPT_USER**.

In the Variable Properties dialog, the **PROMPT_USER Input Node** that was the output in the model has been defined as the **Output**, and the following CellArray has been used to define which variables should be used to supply which **Input Node**. In this instance, clicking in the **Variable Definition** cell gave the list of defined variables from which **DEM** was selected.


Edit Model...
Define Model Input Nodes with Variables:

phic Model: (*.gmd) slope.gmd

Output: n2_PROMPT_U

Layer: (:Layer_1)

Use: Cell Value

Row	Input Node	Node Type	Variable Definition
1	n1_PROMPT_USER	Raster	DEM

Select the spatial model node that represents the calculated variable (Output)

Select variables as inputs to the spatial model in the CellArray here

Thus, a variable has been defined that calculates values on the fly as needed from another variable (in this case, slope is derived from a DEM).

3. Click the Close Model icon, or select **File | Close** from the Spatial Modeler viewer.

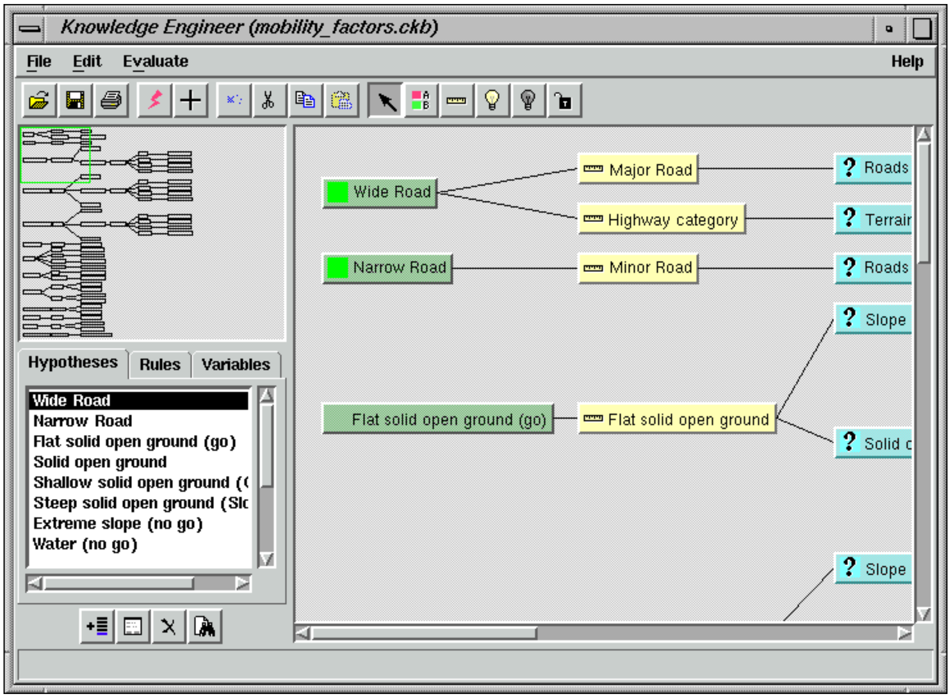
The Spatial Modeler viewer closes.

4. Click **Close** in the Variable Props dialog.

Build Hypotheses

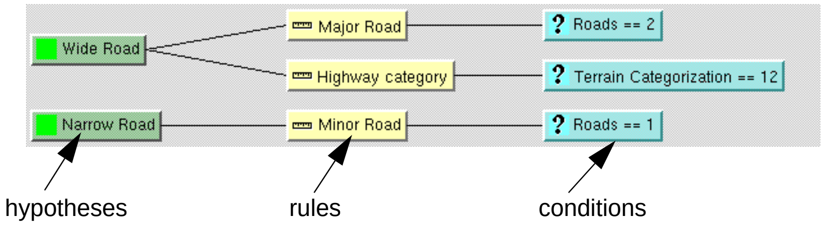
Since you have looked at how the two main types of input variable can be defined, now you can look at how each hypothesis is built in the knowledge base.

The knowledge base **mobility_factors.ckb** displays:


Knowledge Engineer (mobility_factors.ckb)
File
Edit
Evaluate
Help
Hypotheses
Rules
Variables
Wide Road
Narrow Road
Flat solid open ground (go)
Solid open ground
Shallow solid open ground (
Steep solid open ground (Slc
Extreme slope (no go)
Water (no go)
Wide Road
Major Road
Roads
Highway category
Terrair
Narrow Road
Minor Road
Roads
Slope
Flat solid open ground (go)
Flat solid open ground
Solid c
Slope

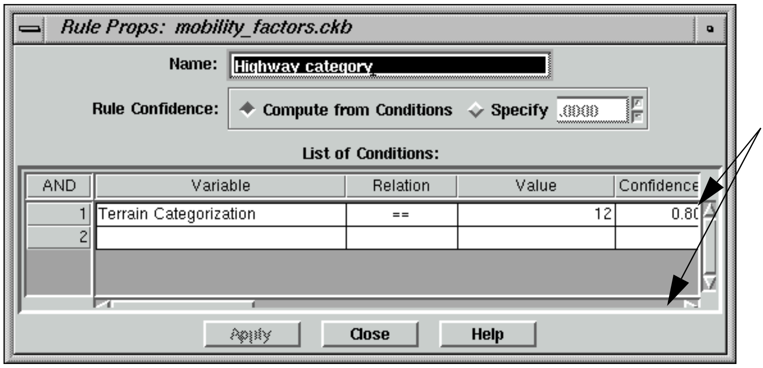
The first two hypotheses, **Wide Road** and **Narrow Road**, are fairly simple. The expert rule in these cases is that something identified as a road is going to be easily traversable by the vehicles, with major roads being better than minor roads.

Consequently, the **Wide Road** hypothesis has two rules (i.e., they each have an **OR** statement so that either needs to be true for the hypothesis to be true). The first looks for major roads (DN value 2) in the **Roads** variable (roads.img), and the second looks for pixels that are potentially identified as roads by the supervised classification.


Wide Road
Major Road
Roads == 2
Highway category
Terrain Categorization == 12
Narrow Road
Minor Road
Roads == 1
hypotheses
rules
conditions

1. Double-click on the **Highway category** rule.

   The Rule Props dialog opens.



Rule Props: mobility_factors.ckb

Name: Highway category

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Variable	Relation	Value	Confidence
1	Terrain Categorization	==	12	0.8
2				

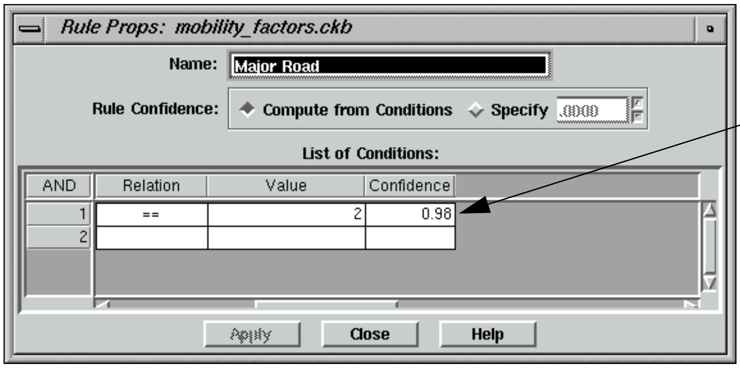
Apply
Close
Help

Use the scroll bar to see the Confidence value in the CellArray

The Rule Props dialog shows how this particular rule depends on the **Terrain Categorization** (**supervised.img**) file.

2. In the Rule Props dialog, click the horizontal scroll bar until you can see the **Confidence** value, **0.80**.
3. In the Knowledge Engineer dialog, click on the **Major Road** rule.

   Its properties display in the Rule Props dialog.
4. Click the horizontal scroll bar until you can see the **Confidence** value, **0.98**.



Rule Props: mobility_factors.ckb

Name: Major Road

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Relation	Value	Confidence
1	==	2	0.98
2			

Apply
Close
Help

The Confidence for the Major Road rule is set higher than that for the Highway category rule

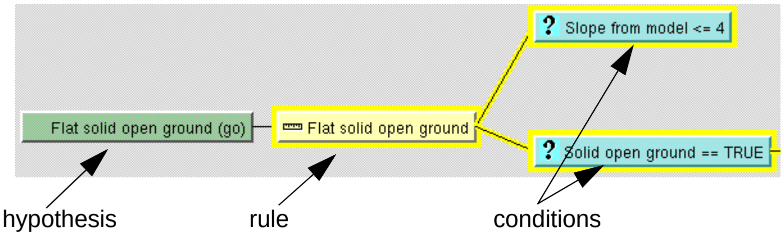
Note that the **Confidence** field for the **Highway category** rule has been set to a much lower value than the **Confidence** for the **Major Road** rule. This is because you are less certain of the results from a maximum likelihood classification than you would be from a road map.

The next four hypotheses work on the same basis. The expert rule is that open ground types are good for vehicle passage. As slopes get steeper, however, the open ground becomes less and less manageable until it becomes impassable at very steep angles.

5. Click **Close** in the Rule Props dialog.

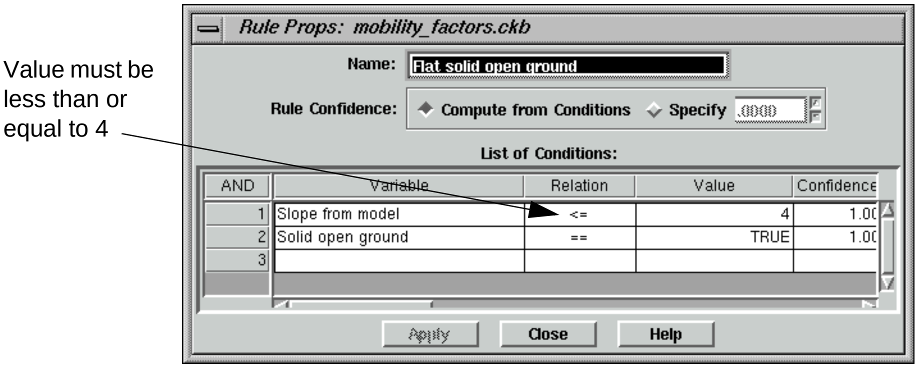
Set ANDing Criteria

These hypotheses also demonstrate the **AND**ing of criteria in a rule. The **Flat solid open ground (go)** hypothesis has only one rule, but that rule has two conditions. Both conditions must be true for a rule to be true.


Flat solid open ground (go)
Flat solid open ground
? Slope from model <= 4
? Solid open ground == TRUE
hypothesis
rule
conditions

1. Double-click the **Slope from model <= 4** condition.

The Rule Props dialog opens.


Value must be less than or equal to 4

Rule Props: mobility_factors.ckb

Name: Flat solid open ground

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Variable	Relation	Value	Confidence
1	Slope from model	<=	4	1.0
2	Solid open ground	==	TRUE	1.0
3				

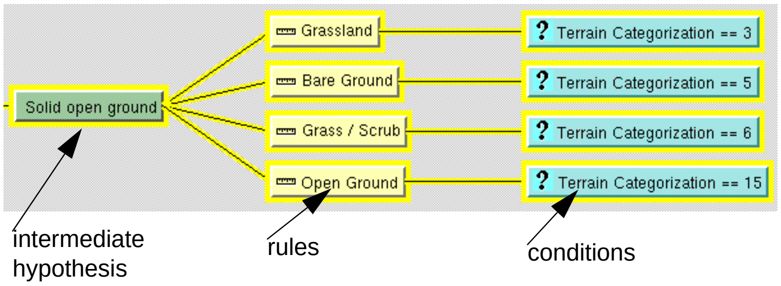
Apply
Close
Help

In the **List of Conditions** CellArray, the slope calculated by the **Slope from model** variable must be less than or equal to **4** degrees, and (i.e., **AND**, note the far left column of the CellArray) the **Solid open ground** variable must be true before this rule can be true.

2. Click **Close** in the Rule Props dialog.

   The rule **Flat solid open ground** is defined by an intermediate hypothesis: **Solid open ground**.

3. In the Knowledge Engineer dialog, click the horizontal scroll bar and move it to the right until you can see the intermediate hypothesis **Solid open ground**.

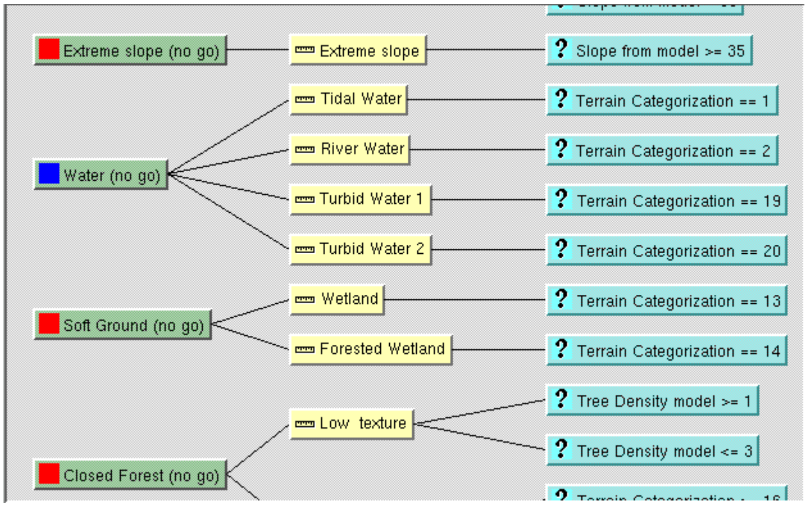

Solid open ground
Grassland
Bare Ground
Grass / Scrub
Open Ground
Terrain Categorization == 3
Terrain Categorization == 5
Terrain Categorization == 6
Terrain Categorization == 15
intermediate hypothesis
rules
conditions

The intermediate hypothesis **Solid open ground** uses the **Terrain Categorization** variable to identify various types of open ground cover.

Check Other Hypotheses

You can check the other hypotheses listed along the left side of the Knowledge Engineer dialog.

1. Position the horizontal scroll bar to the extreme left in the Knowledge Engineer dialog.
2. Move the vertical scroll bar down until you see the **Water (no go)** hypothesis.


Extreme slope (no go)
Extreme slope
Slope from model >= 35
Tidal Water
Terrain Categorization == 1
Water (no go)
River Water
Terrain Categorization == 2
Turbid Water 1
Terrain Categorization == 19
Turbid Water 2
Terrain Categorization == 20
Wetland
Terrain Categorization == 13
Soft Ground (no go)
Forested Wetland
Terrain Categorization == 14
Low texture
Tree Density model >= 1
Tree Density model <= 3
Closed Forest (no go)

The **Water (no go)** hypothesis is a simple expert rule that states that if the **Terrain Categorization** variable shows that a location is water, it is impassable.

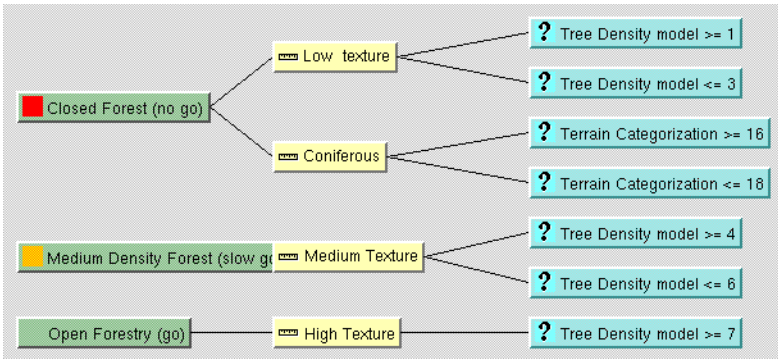
3. Note the hypothesis below **Water (no go)**: **Soft Ground (no go)**.

Similarly, the **Soft Ground (no go)** hypothesis is a simple expert rule that states that if the **Terrain Categorization** variable shows that a location is a type of wetland, it is impassable.

Introduce Spatial Logic to the Knowledge Base

The three forest hypotheses are more complicated, and show how spatial operators can be used within the IMAGINE Expert Classifier to create a spatially aware classifier (rather than a traditional per-pixel classifier).

1. Use the vertical scroll bar to scroll down to the various forest hypotheses.

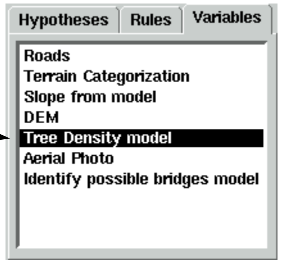

Low texture
Tree Density model >= 1
Tree Density model <= 3
Closed Forest (no go)
Coniferous
Terrain Categorization >= 16
Terrain Categorization <= 18
Tree Density model >= 4
Medium Density Forest (slow g
Medium Texture
Tree Density model <= 6
Open Forestry (go)
High Texture
Tree Density model >= 7

The expert rule is that open forest canopies are easier to traverse than closed forest canopies because space between trunks is not as large with a closed canopy (i.e., trees are older and have large trunks). However, it is difficult to determine canopy closure on a per-pixel basis.

To a certain degree, canopy closure can be inferred from the type of tree cover. For instance, if the cover is coniferous, then it is more likely that you have a close canopy and close trunks. However, this is more difficult to state for mixed and other types of tree cover.

Consequently, this particular hypothesis also looks at panchromatic aerial photography to try and determine crown closure. However, you cannot determine crown state on a per-pixel basis. Instead, the **Tree Density model** variable (using the **Aerial Photo** and **Terrain Categorization** variables as inputs) executes a graphical model (**texture.gmd**).

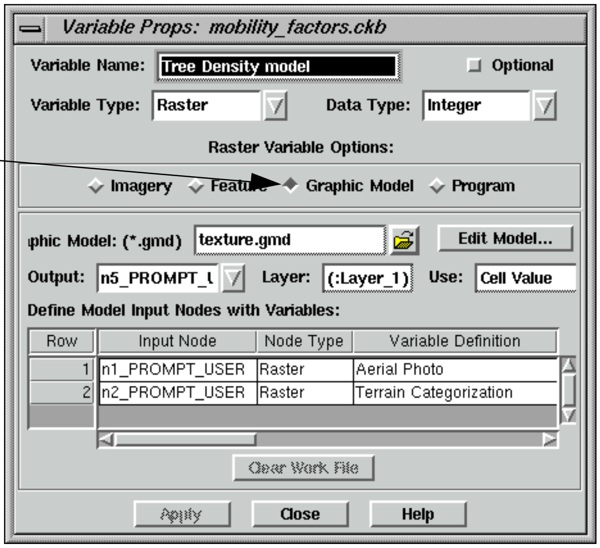
2. Click the **Variables** tab in the Knowledge Engineer dialog.


Hypotheses
Rules
Variables
Roads
Terrain Categorization
Slope from model
DEM
Tree Density model
Aerial Photo
Identify possible bridges model

Double-click Tree Density model

3. Double-click to select the **Tree Density model** variable.

The Variable Props dialog for the **Tree Density model** variable opens:


Variable Props: mobility_factors.ckb
Optional
Raster Variable Options:
Imagery
Feature
Graphic Model
Program
Edit Model...
Define Model Input Nodes with Variables:
Clear Work File
Apply
Close
Help

Variable Name: Tree Density model

Variable Type: Raster

Data Type: Integer

phic Model: (*.gmd) texture.gmd

Output: n5_PROMPT_U

Layer: (:Layer_1)

Use: Cell Value

Row	Input Node	Node Type	Variable Definition
1	n1_PROMPT_USER	Raster	Aerial Photo
2	n2_PROMPT_USER	Raster	Terrain Categorization

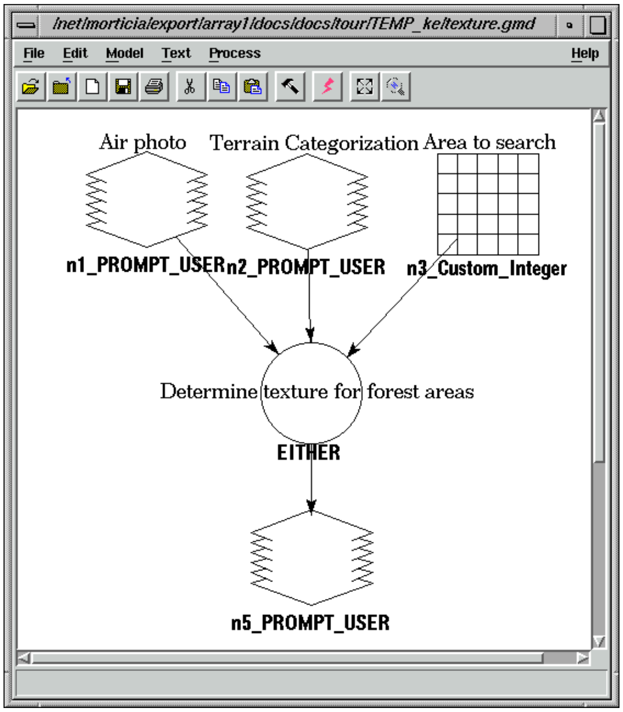
Graphic Model is selected for the variable option

4. Note that the **Raster Variable Options** for the **Tree Density model** variable is set to **Graphic Model**.

The model that provides information for the **Tree Density model** variable is **texture.gmd**.

5. Click the **Edit Model** button in the Variable Props dialog.

The Spatial Modeler viewer opens, which displays the model **texture.gmd**.


/net/morticia/export/array1/docs/docs/tour/TEMP_ke/texture.gmd
File
Edit
Model
Text
Process
Help
Air photo
Terrain Categorization
Area to search
n1_PROMPT_USER
n2_PROMPT_USER
n3_Custom_Integer
Determine texture for forest areas
EITHER
n5_PROMPT_USER

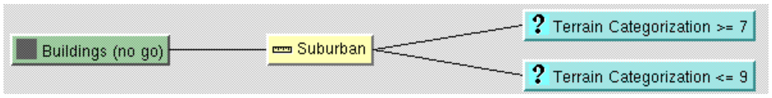
This model, **texture.gmd**, is a model that looks at every location that has been determined to be tree cover (from the **Terrain Categorization** variable), and analyzes both the air photo DN value at that location and the DN values in surrounding locations (using a 3 × 3 moving window) to determine local texture. This spatial variable is then used in the **Close Forest (no go)**, **Medium Density forest (slow go)**, and **Open Forestry (go)** hypotheses to determine the relative density of the tree canopy, and thereby determine the relative ease of vehicular passage.

6. Select **File | Close** from the Spatial Modeler viewer.

7. Click **Close** in the Variable Props dialog displaying properties for the **Tree Density model** variable.

Check Buildings Hypothesis

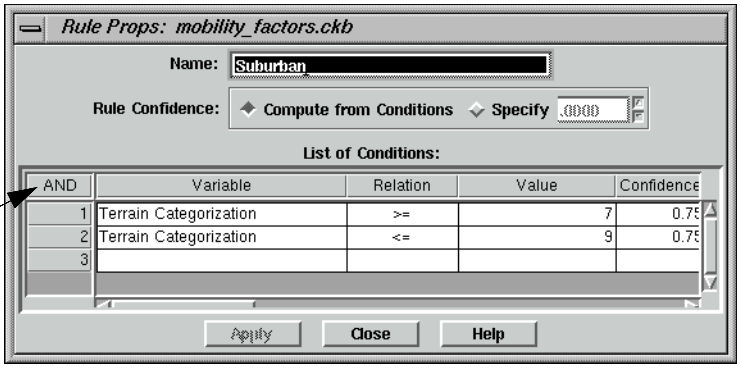
The **Buildings (no go)** hypothesis is a simple expert rule that states that if the **Terrain Categorization** shows that a location is a type of urban area, it is impassable.


Buildings (no go)
Suburban
Terrain Categorization >= 7
Terrain Categorization <= 9

Confidences are kept low on these rules so that they do not override the **Wide Road** and **Narrow Road** hypotheses. That is, roads within urban areas are still traversable.

1. Double-click on the **Suburban** rule attached to the **Buildings (no go)** hypothesis.

The Rule Props for the **Suburban** rule open.



Rule Props: mobility_factors.ckb

Name: Suburban

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Variable	Relation	Value	Confidence
1	Terrain Categorization	>=	7	0.7
2	Terrain Categorization	<=	9	0.7
3				

This rule uses an AND statement

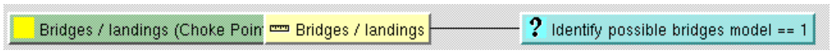
2. In the Rule Props dialog, move the horizontal scroll bar all the way to the right.

3. Notice that the **Confidence** values are set to **0.75**.

4. Click **Close** in the Rule Props dialog.

Identify Choke Points

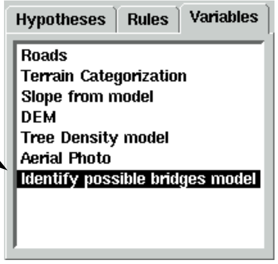
The final hypothesis is another good example of spatially enabling the IMAGINE Expert Classifier. This hypothesis identifies choke points in the road network—points where the road narrows considerably and traffic cannot circumnavigate, thereby representing a potential no go point. The main example of this is bridges.



Bridges / landings (Choke Poin
Bridges / landings
Identify possible bridges model == 1

Identification of bridges might sound like an easy proposition: find roads that are on water. However, the only information we have on location of water bodies is from the landcover classification (the **Terrain Categorization** variable), which cannot identify water that flows below other features. Consequently, a more complex approach is required.

1. Click the **Variables** tab in the Knowledge Engineer dialog.

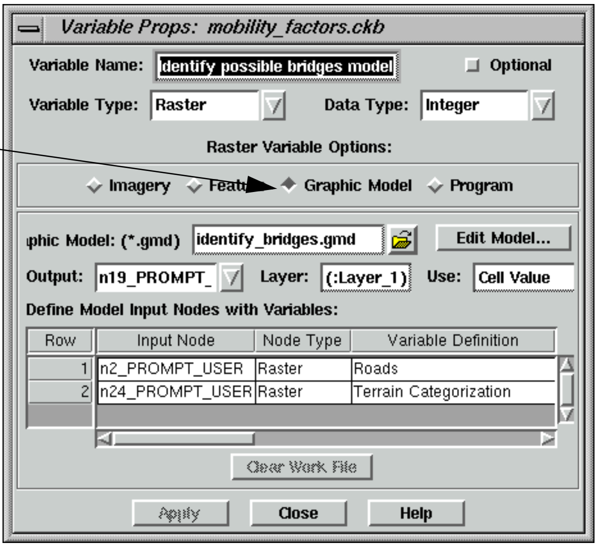


Roads
Terrain Categorization
Slope from model
DEM
Tree Density model
Aerial Photo
Identify possible bridges model

Double-click the Identify possible bridges model variable

2. In the **Variables** tab, double-click **Identify possible bridges model**.

The Variable Props dialog for the variable **Identify possible bridges model** opens.



This variable also uses a Graphic Model

Variable Props: mobility_factors.ckb

Variable Name: identify possible bridges model

☐ Optional

Variable Type: Raster

Data Type: Integer

Raster Variable Options:

◇ Imagery ◇ Featu ◆ Graphic Model ◇ Program

phic Model: (*.gmd) identify_bridges.gmd

Output: n19_PROMPT_ Layer: (:Layer_1) Use: Cell Value

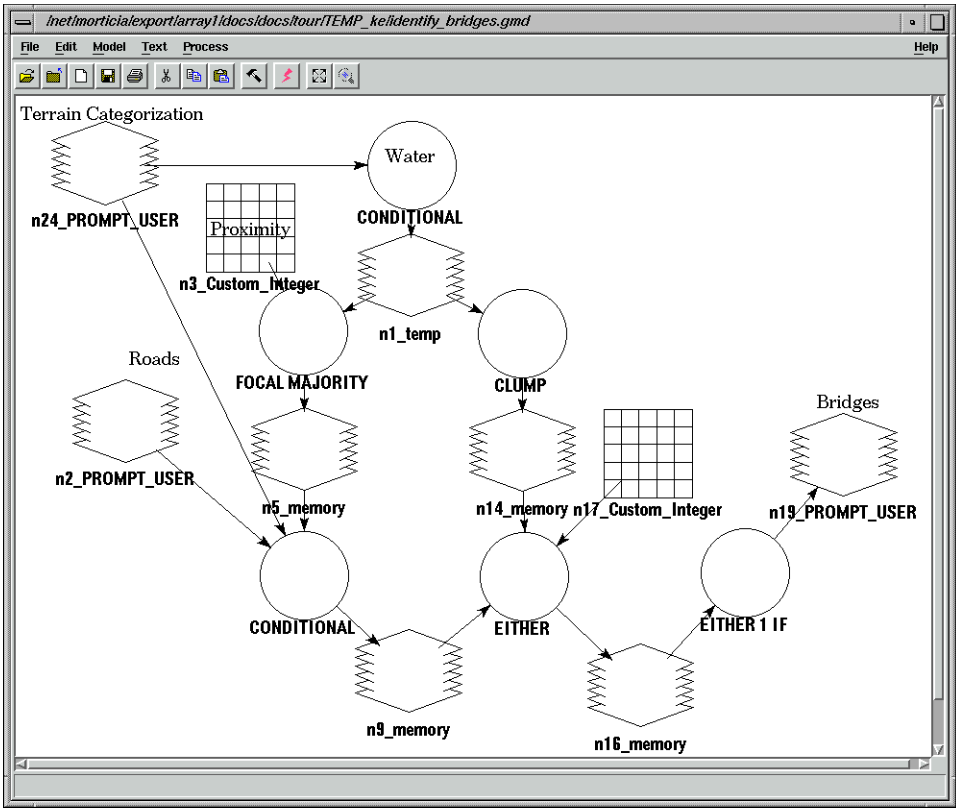
Define Model Input Nodes with Variables:

Row	Input Node	Node Type	Variable Definition
1	n2_PROMPT_USER	Raster	Roads
2	n24_PROMPT_USER	Raster	Terrain Categorization

Edit Model...
Clear Work File
Apply
Close
Help

3. Click the **Edit Model** button in the Variable Props dialog.

The model used to identify potential bridges, **identify_bridges.gmd**, shows the expert rule.



/net/morticia/export/array1/docs/docs/tour/TEMP_ke/identify_bridges.gmd
File
Edit
Model
Text
Process
Help
Terrain Categorization
n24_PROMPT_USER
Water
CONDITIONAL
Proximity
n3_Custom_Integer
n1_temp
FOCAL MAJORITY
CLUMP
Roads
n2_PROMPT_USER
n5_memory
n14_memory
n17_Custom_Integer
Bridges
n19_PROMPT_USER
CONDITIONAL
EITHER
EITHER 1 IF
n9_memory
n16_memory

Since you cannot immediately identify roads over water, you must instead look at roads in close proximity to water. This could be done by buffering (performing a Search function) on the roads and overlaying this with the location of water pixels. However, many roads simply run alongside lakes or rivers and do not necessarily therefore constitute a choke hazard. Instead, it is better to identify roads that occur in close proximity to at least two discrete water bodies (i.e., one on either side of the bridge).

Therefore, **identify_bridges.gmd** first identifies all water pixels from the landcover classification. These locations are fed into two processes. The first finds all locations that are in close proximity to water by using a 5 × 5 circular moving window). These are then overlain with road locations (from **Roads** and **Terrain Categorization** variables) to identify roads in close proximity to water. At the same time, the water pixels are run through a Clump process to produce uniquely numbered, discrete water bodies. A Focal Diversity function is then used at each location determined to be a road in close proximity to water to determine how many of these discrete water bodies are close by. If more than two water bodies are identified, then that road is flagged as being a potential bridge or other choke point. This information is then used in the **Bridges/landings (Choke Point)** expert rule.

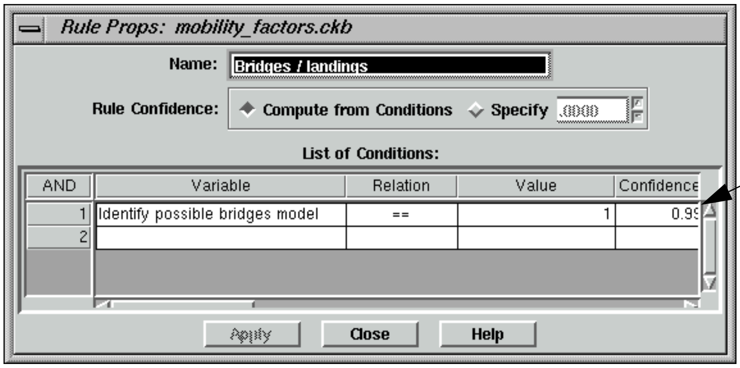
This provides a clear example of how the IMAGINE Expert Classifier can be used to integrate spatially aware rules. In this case, the values of neighboring pixels are analyzed to help determine the land use (bridge as opposed to road) of the target pixels.

4. Click the Close Model icon, or select **File | Close** from the Spatial Modeler viewer.

Also note that the **Bridges/landings (Choke Point)** hypothesis is always going to occur at pixel locations that have also met the requirements to be in the **Wide Road** or **Narrow Road** classes (it is extremely difficult to create expert rules that are always mutually exclusive). Consequently, the **Confidence** values on the **Bridges/landings** rule have been set higher than those for the normal road rules. In this way, the **Bridges/landings (Choke Point)** hypothesis always takes precedence in the classifications.

5. In the Knowledge Engineer dialog, double-click the **Bridges/landings** rule.

The Rule Props dialog for **Bridges/landings** opens.



Rule Props: mobility_factors.ckb

Name: Bridges / landings

Rule Confidence: ◆ Compute from Conditions ◇ Specify .0000

List of Conditions:

AND	Variable	Relation	Value	Confidence
1	Identify possible bridges model	==	1	0.9
2				

Check the Confidence value

6. Move the horizontal scroll bar to the right see the **Confidence** value.

7. Note that the **Confidence** of the variable **Identify possible bridges model** is set to **0.99**.

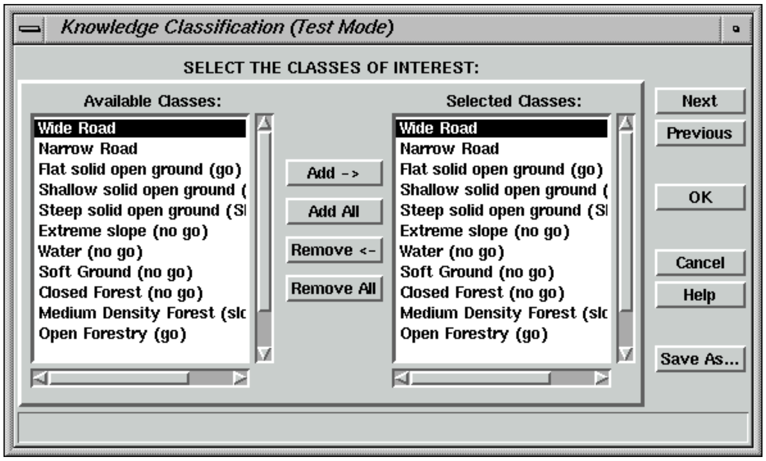
8. Click **Close** in the Rule Props dialog.

9. Click **Close** in the Variable Props dialog.

Run the Expert Classification

1. In the Knowledge Engineer dialog, click the Run icon, or select Evaluate | Test Knowledge Base.

The Knowledge Classification (Test Mode) dialog opens on the **Select the Classes of Interest** panel.



Knowledge Classification (Test Mode)

SELECT THE CLASSES OF INTEREST:

Available Classes:

Wide Road
Narrow Road
Flat solid open ground (go)
Shallow solid open ground (
Steep solid open ground (Sl
Extreme slope (no go)
Water (no go)
Soft Ground (no go)
Closed Forest (no go)
Medium Density Forest (slc
Open Forestry (go)

Selected Classes:

Wide Road
Narrow Road
Flat solid open ground (go)
Shallow solid open ground (
Steep solid open ground (Sl
Extreme slope (no go)
Water (no go)
Soft Ground (no go)
Closed Forest (no go)
Medium Density Forest (slc
Open Forestry (go)

You want to see results for all of the classes; therefore, you can proceed to the next panel.

2. Click the **Next** button in the Knowledge Classification (Test Mode) dialog.

The **Select the Input Data for Classification** panel opens.

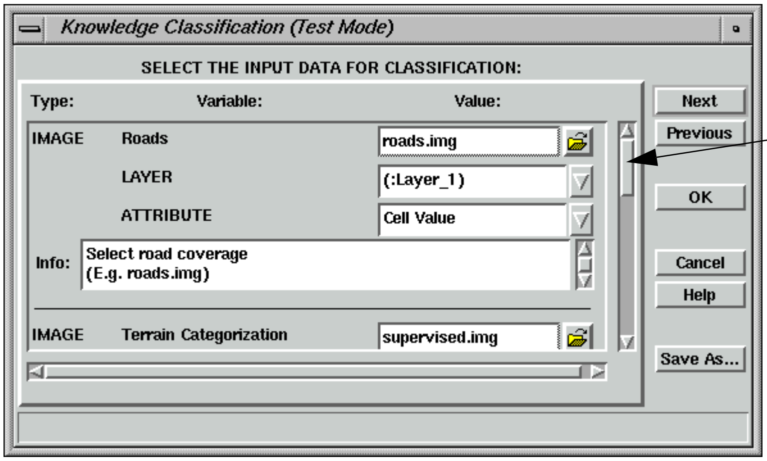

Knowledge Classification (Test Mode)
SELECT THE INPUT DATA FOR CLASSIFICATION:
Type:
Variable:
Value:
Next
Previous
OK
Cancel
Help
Save As...

IMAGE Roads: roads.img

LAYER: (:Layer_1)

ATTRIBUTE: Cell Value

Info: Select road coverage
(E.g. roads.img)

IMAGE Terrain Categorization: supervised.img

Use the scroll bar to see the values assigned to variables

This panel enables you to identify the files to be used as variables, which were set to the **Leave Undefined (Prompt Analyst)** state.

3. Use the vertical scroll bar to see the variables and their corresponding files.

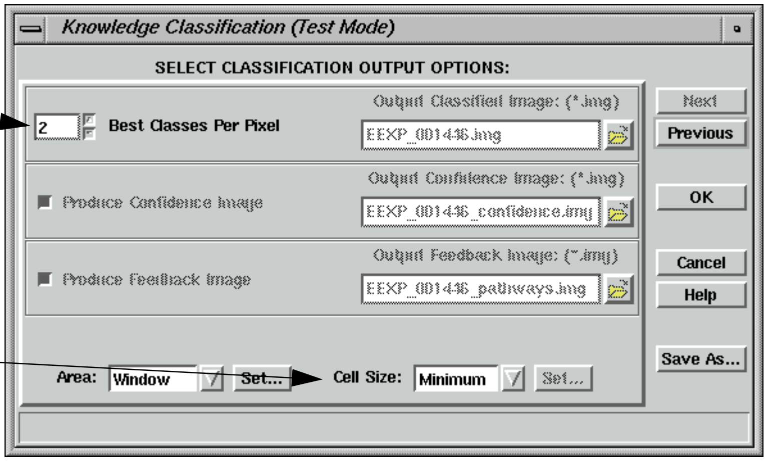
   In this Knowledge Base, the **Roads** variable is associated with **roads.img**, the **Terrain Categorization** variable is associated with **supervised.img**, the **DEM** variable is associated with **30meter.img**, and the **Aerial Photo** variable is associated with **mason_ap.img**.

4. Click **Next** in the Knowledge Classification (Test Mode) dialog.

   The **Select Classification Output Options** panel opens.

Change the Best Classes Per Pixel value to 2

Confirm that the Cell Size is set to Minimum


Knowledge Classification (Test Mode)
SELECT CLASSIFICATION OUTPUT OPTIONS:
Next
Previous
OK
Cancel
Help
Save As...
Set...
Set...

Best Classes Per Pixel: 2

Output Classified Image: (*.img) EEXP_001446.img

☐ Produce Confidence Image — Output Confidence Image: (*.img) EEXP_001446_confidence.img

☐ Produce Feedback Image — Output Feedback Image: (*.img) EEXP_001446_pathways.img

Area: Window

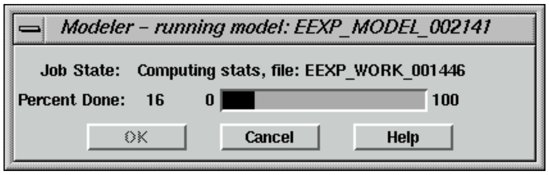
Cell Size: Minimum

5. Change the **Best Classes Per Pixel** value to **2**.

6. Confirm that the **Cell Size** is set to **Minimum**.

7. Click **OK** in the **Select Classification Output Options** panel of the Knowledge Classification (Test Mode) dialog.

Job Status dialogs open, tracking the progress of the expert classification.



Modeler – running model: EEXP_MODEL_002141

Job State: Computing stats, file: EEXP_WORK_001446

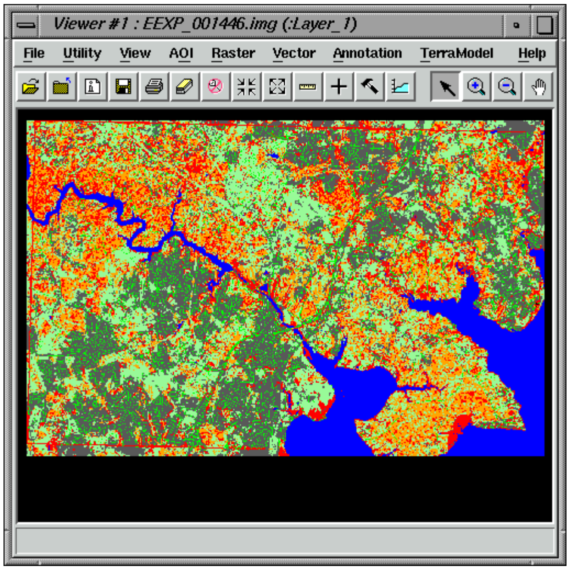
Percent Done: 16

0
100
OK
Cancel
Help

8. When the job is complete, click **OK** in the Job Status dialogs.

You can set a preference to automatically close the Job Status dialog after computation is complete. It is located in the **User Interface & Session** *category of the Preference Editor.*

When the process is complete, the classification displays in a Viewer.



Viewer #1 : EEXP_001446.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

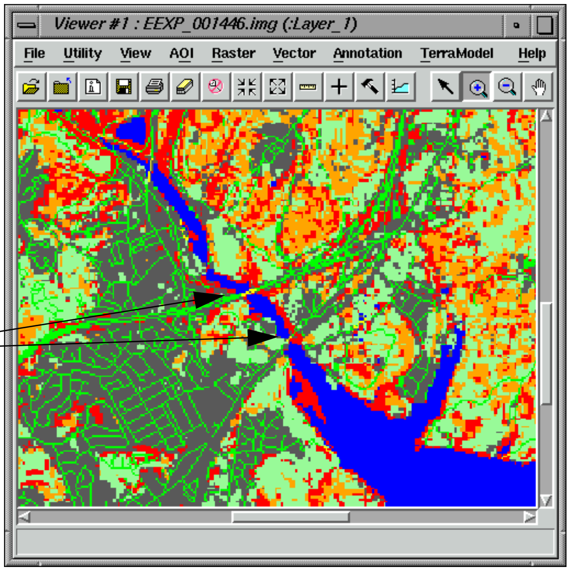
Evaluate River Areas

Now that the classification is complete, you should zoom in and see what the IMAGINE Expert Classifier designated as potential bridges.

1. In the Viewer tool bar, click the Zoom In icon.

2. Move your mouse into the Viewer, and click on an area of the river.

3. Click as many times as necessary in order to see the detail of the area.


Viewer #1 : EEXP_001446.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Two bridges are located in this area of the image

4. Zoom in further until you can see yellow pixels at bridge locations, which indicate the **Bridges/landings (Choke Point)** class.

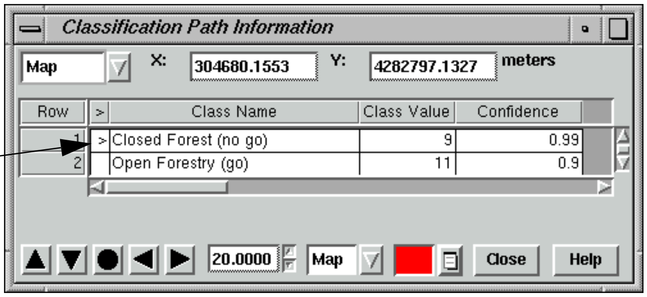
If you refer back to the Knowledge Engineer dialog, you can see that the **Bridges/landings (Choke Point)** *hypotheses has a yellow color square. Therefore, pixels in that class are also yellow.*

Use Pathway Feedback

You can use the pathway feedback cursor to analyze the classification in the Viewer.

1. Click the Classification Pathway Feedback Mode icon in the Knowledge Engineer dialog.

The Classification Path Information dialog opens.


Classification Path Information
Close
Help

Map X: 304680.1553 Y: 4282797.1327 meters

Row	>	Class Name	Class Value	Confidence
1	>	Closed Forest (no go)	9	0.99
2		Open Forestry (go)	11	0.9

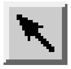
20.0000 Map

The class beneath the inquire cursor is identified here

In the Classification Path Information dialog, the second row in the CellArray specifies the second most likely class (hypothesis) for this pixel (since you requested the **2 Best Classes Per Pixel** on page 592).

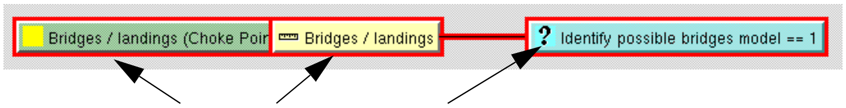
An inquire cursor is placed in the Viewer containing the classification, and the pathway it corresponds to is highlighted in red in the Knowledge Engineer dialog.

2. Click the Select icon from the Viewer tool bar.



3. Using your mouse click, hold, and drag the inquire cursor to a yellow pixel in the Viewer.

The Classification Path Information dialog and the Knowledge Engineer dialog update accordingly.


Bridges / landings (Choke Poir
Bridges / landings
Identify possible bridges model == 1

The hypothesis, rule, and condition are outlined in red

4. Continue to move the inquire cursor around in the Viewer, and analyze the results in the Classification Path Information dialog and the Knowledge Engineer dialog.

5. When you are finished, click **Close** in the Classification Path Information dialog.

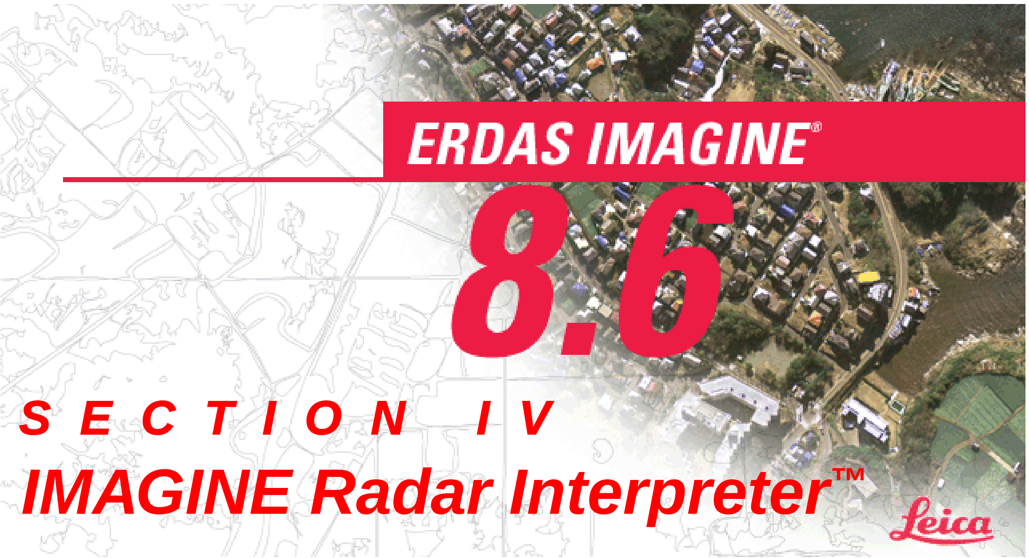
---

*A graphical model (***clean_up_mobility.gmd***) is supplied for removing the salt and pepper classification pixels from the final land-use map. This model uses a focal majority, but avoids altering the road and water classes.*

Select **File | Close** in the Viewer containing the classification.

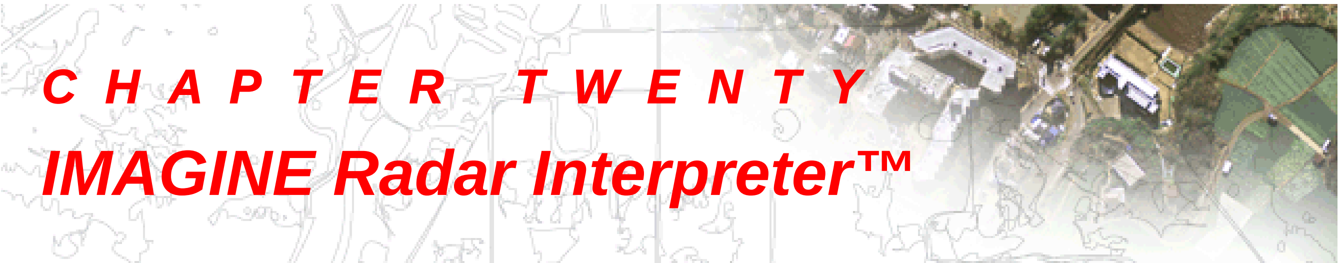
Select **File | Close** in the Knowledge Engineer dialog.

The knowledge base **mobility_factors.ckb** is an example of how a knowledge base can be built to take into account spatial rather than (or as well as) spectral per-pixel relationships to derive land use information. It also shows how commonly repeated tasks can be automated for repeating within an organization, or to repeat the same methodology at other organizations. instead of running several separate spatial models and trying to integrate the results, the entire process is captured in one knowledge base that can be easily applied to other data and other locations, with consistent results.


ERDAS IMAGINE®
8.6
Leica

SECTION IV

IMAGINE Radar Interpreter™



CHAPTER TWENTY

IMAGINE Radar Interpreter™

Introduction

Building a model, creating a map, or rectifying an image requires certain steps, regardless of the data you are using. However, processing radar data is application-driven, so there is no preset path to follow. Therefore, this tour guide shows you how the functions work, but you have to experiment on your own data files for your own applications.

The default settings in the IMAGINE Radar Interpreter module dialogs provide acceptable results. However, it is recommended that you experiment with the settings to obtain the best results.

NOTE: The data used in this tour guide are in the <IMAGINE_HOME>/examples directory. Replace <IMAGINE_HOME> with the name of the directory where ERDAS IMAGINE is installed on your system.

Although you can use the IMAGINE Radar Interpreter functions in any order, it is recommended that you follow this tour guide in the order that it is presented. It is important to address speckle noise before any other processing.

See "Chapter 5: Enhancement" *in the* ERDAS Field Guide *for more theoretical information about using the Radar module.*

The IMAGINE Radar Interpreter is part of the IMAGINE Radar Mapping Suite™, which also includes IMAGINE OrthoRadar™, IMAGINE StereoSAR DEM™, IMAGINE IFSAR DEM™, and the Generic SAR Node.

Approximate completion time for this tour guide is 45 minutes.

Suppress Speckle Noise

In this section, you display two images—one that has been despeckled, and one raw radar image. The objective is to make the two images look alike by using the Speckle Suppression function.

With all speckle suppression filters there is a trade-off between noise reduction and loss of resolution. Each data set and each application has a different acceptable balance between these two factors. The IMAGINE Radar Interpreter module Speckle Suppression filters have been designed to be versatile and gentle in reducing noise (and resolution).

In this section, you also calculate the coefficient of variation for an image. This variable is required to fine-tune many Speckle Suppression filters.

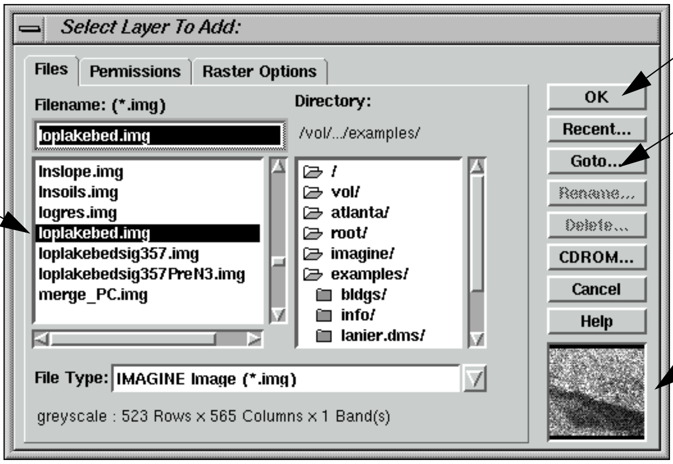
When processing radar imagery, it is very important to use the Speckle Suppression functions before other image processing functions to avoid incorporating speckle into the image.

Preparation

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer menu bar, select **File | Open | Raster Layer** or click the Open icon in the tool bar.

The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Permissions
Raster Options
Filename: (*.img)
Directory:
loplakebed.img
/vol/.../examples/
lnslope.img
lnsoils.img
logres.img
loplakebed.img
loplakebedsig357.img
loplakebedsig357PreN3.img
merge_PC.img
/
vol/
atlanta/
root/
imagine/
examples/
bldgs/
info/
lanier.dms/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help
File Type:
IMAGINE Image (*.img)
greyscale : 523 Rows x 565 Columns x 1 Band(s)

Click here to select the file

Click here to display the file

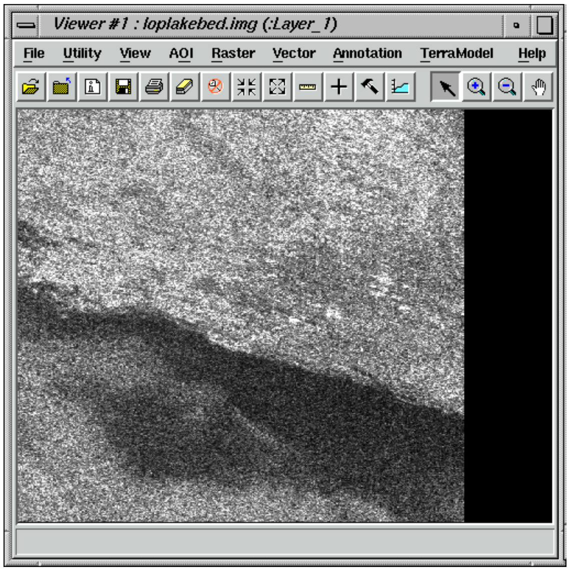
Click here to change the directory, if needed

A preview of the image displays here

2. In the Select Layer To Add dialog under **Filename**, click on **loplakebed.img**.

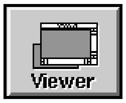
3. Click **OK** in the Select Layer To Add dialog.

The **loplakebed.img** file is displayed in the Viewer.


Viewer #1 : loplakebed.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

This image is a subset from imagery taken by the Shuttle Imaging Radar (SIR-A) experiment. It is L-band with 25 m pixels. This scene is the shore of Lop Nor Lake in the Xinjiang Province, Peoples' Republic of China. This is an area of desiccated overflow basins surrounded by a series of parallel, wind-scoured, sedimentary ridges. The speckle in this image is obvious.

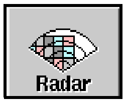
4. In the ERDAS IMAGINE icon panel, click on the Viewer icon to open another Viewer.



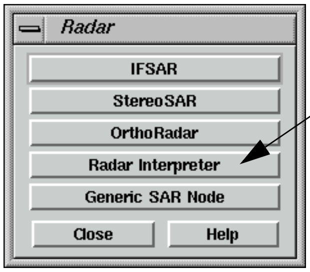
5. From the ERDAS IMAGINE menu bar, select **Session | Tile Viewers** to position and size the Viewers so that you can see the side-by-side Viewers on the screen.

   This helps you to view and evaluate the resultant image after each filter pass, and then decide if another pass is needed to obtain the desired results.

6. Click on the Radar icon on the ERDAS IMAGINE icon panel.



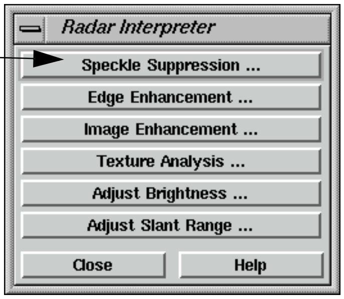
The **Radar** menu opens.


Radar
IFSAR
StereoSAR
OrthoRadar
Radar Interpreter
Generic SAR Node
Close
Help

Click Radar Interpreter

7. In the **Radar** menu, click **Radar Interpreter**.

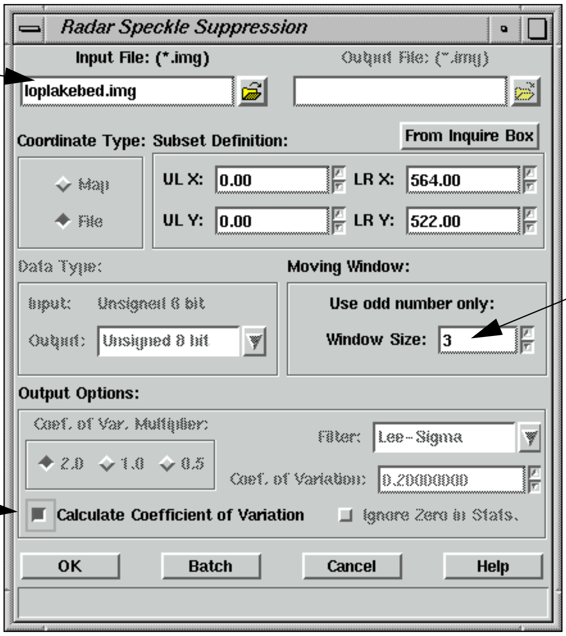
   The **Radar Interpreter** menu opens.


Radar Interpreter
Speckle Suppression ...
Edge Enhancement ...
Image Enhancement ...
Texture Analysis ...
Adjust Brightness ...
Adjust Slant Range ...
Close
Help

Select Speckle Suppression

8. In the **Radar Interpreter** menu, select **Speckle Suppression**.

   The Radar Speckle Suppression dialog opens.


Radar Speckle Suppression
From Inquire Box
OK
Batch
Cancel
Help

Input File: (*.img) loplakebed.img
Output File: (*.img)
Coordinate Type: ○ Map ● File
Subset Definition:
UL X: 0.00
LR X: 564.00
UL Y: 0.00
LR Y: 522.00
Data Type:
Input: Unsigned 8 bit
Output: Unsigned 8 bit
Moving Window:
Use odd number only:
Window Size: 3
Output Options:
Coef. of Var. Multiplier: ● 2.0 ○ 1.0 ○ 0.5
Filter: Lee-Sigma
Coef. of Variation: 0.20000000
☒ Calculate Coefficient of Variation
☐ Ignore Zero in Stats.

Enter the name of the input file here

Enter window size here (windows are always square)

Click here to calculate the Coefficient of Variation

9. In the Radar Speckle Suppression dialog under **Input File**, enter the file **loplakebed.img**.

Calculate Coefficient of Variation

Next, you calculate the coefficient of variation to be used in this function.

Coefficient of Variation

The coefficient of variation, as a scene-derived parameter, is a necessary input parameter for many of the filters. (It is also useful in evaluating and modifying VIS/IR data for input to a 4-band composite image or in preparing a 3-band ratio color composite.)

Speckle in imaging radar can be mathematically modeled as multiplicative noise with a mean of 1. The standard deviation of the noise can be mathematically defined as:

Standard Deviation of the noise => $\frac{\sqrt{VARIANCE}}{MEAN}$ = Coefficient of Variation

It is assumed that imaging radar speckle noise follows a Rayleigh distribution. This yields a theoretical value for standard deviation (SD) of .52 for 1-look radar data and
SD = .26 for 4-look radar data.

The following table gives theoretical coefficient of variation values for various look-averaged radar scenes.

Table 20-1: Coefficient of Variation Values for Look-averaged Radar Scenes

Number of Looks (scenes)	Coefficient of Variation Value
1	.52
2	.37
3	.30
4	.26
6	.21
8	.18

1. In the Radar Speckle Suppression dialog, click the checkbox for **Calculate Coefficient of Variation**.

   All the other options in the dialog are disabled, except for the **Subset Definition** and **Moving Window**. If desired, you could specify a subset area of the image for which to calculate the coefficient of variation.

2. Under **Moving Windows**, confirm that the **Window Size** is set to **3**.

3. Click **OK** in the Radar Speckle Suppression dialog.

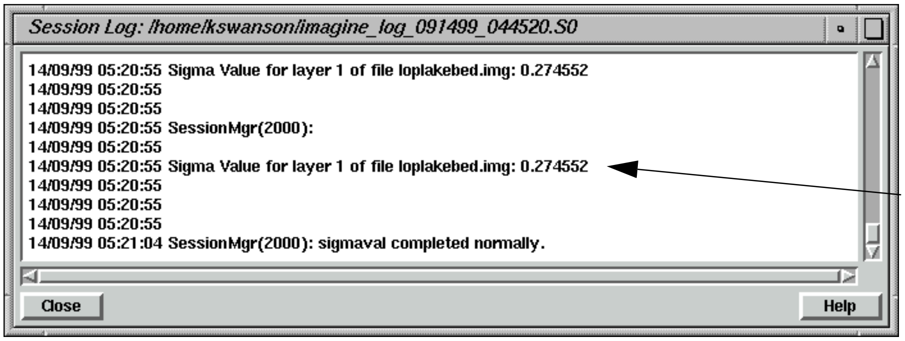
The Radar Speckle Suppression dialog closes and a Job Status dialog displays, indicating the progress of the function.

4. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

Depending on your eml **Preferences** *(under* **Session | Preferences | User Interface & Session | Keep Job Status Box***), when the Job Status bar shows 100 (indicating that the job is 100% done), you must either click* **OK** *to close the dialog or the dialog closes automatically.*

5. If it is not already displayed, open the Session Log by selecting **Session | Session Log** from the ERDAS IMAGINE menu bar.

The calculated coefficient of variation is reported in the **Session Log**, as shown in the following example.



```
14/09/99 05:20:55 Sigma Value for layer 1 of file loplakebed.img: 0.274552
14/09/99 05:20:55
14/09/99 05:20:55
14/09/99 05:20:55 SessionMgr(2000):
14/09/99 05:20:55
14/09/99 05:20:55 Sigma Value for layer 1 of file loplakebed.img: 0.274552
14/09/99 05:20:55
14/09/99 05:20:55
14/09/99 05:20:55
14/09/99 05:21:04 SessionMgr(2000): sigmaval completed normally.
```

Calculated Coefficient of Variation is reported here

When using the filters in the Speckle Suppression function, you should calculate the coefficient of variation for the input image and use a number close to the calculated coefficient of variation for optimum results.

6. Click **Close** in the Session Log.

Run Speckle Suppression Function

1. In the **Radar Interpreter** menu, select **Speckle Suppression**.

   The Radar Speckle Suppression dialog opens.

2. Under **Input File**, enter the file name **loplakebed.img**.
3. Under **Output File**, enter **despeckle1.img** in the directory of your choice.

NOTE: Be sure to remember the directory where you have saved the output file. This is important when you display the output file in a Viewer.

4. Under **Coef. of Var. Multiplier** (under **Output Options**), click **0.5**.

5. Under **Output Options**, confirm **Lee-Sigma** is selected from the popup list next to **Filter**.

6. Under **Output Options**, enter **.275** for the **Coef. of Variation** (coefficient of variation), then press Return on your keyboard.

   This is the value (.275) that was reported in the Session Log when you calculated the coefficient of variation.

7. Click **OK** in the Radar Speckle Suppression dialog.

   The Radar Speckle Suppression dialog closes and a Job Status dialog displays, indicating the progress of the function.

8. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of **Viewer #2**, select **File | Open | Raster Layer** or click the Open icon 📂 in the tool bar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select **despeckle1.img** as the file to open and click **OK**.

3. Repeat steps (1. through (7. under "Run Speckle Suppression Function" to apply the Speckle Suppression function iteratively to the output images, using the following parameters for passes 2 and 3.

Table 20-2: Speckle Suppression Parameters

Pass	Input file	Output file	Coef. of Var.	Coef. of Var. Multiplier	Window Size
1	loplakebed.img	despeckle1.img	0.275	0.5	3 × 3
2	despeckle1.img	despeckle2.img	0.195	1	5 × 5
3	despeckle2.img	despeckle3.img	0.103	2	7 × 7

You MUST enter a new output file name each time you run a speckle suppression filter. In this exercise, name each pass sequentially (e.g., **despeckle1.img**, **despeckle2.img**, **despeckle3.img**, *etc.).*

Speckle Suppression Filters

The Speckle Suppression filters can be used repeatedly in as many passes as needed.

Similarly, there is no reason why successive passes must be done with the same filter.

The following filtering sequence might be useful prior to a classification.

Table 20-3: Filtering Sequence

Filter	Pass	Sigma Value	Sigma Multiplier	Window
Lee	1	0.26	NA	3 × 3
Lee	2	0.22	NA	5 × 5
Local Region	3	NA	NA	7 × 7 or 9 × 9

Use Histograms to Evaluate Images

Next, the ImageInfo method of histogram display is explained.

Histograms

Viewing the histograms of an image is often helpful in determining: the need for filtering, the type of filter to use, and the results of filtering.

You can see a histogram of an image through:

- **Tools | Image Information | View | Histogram** from the ERDAS IMAGINE menu bar
- **Utility | Layer Info | View | Histogram** from the Viewer menu bar

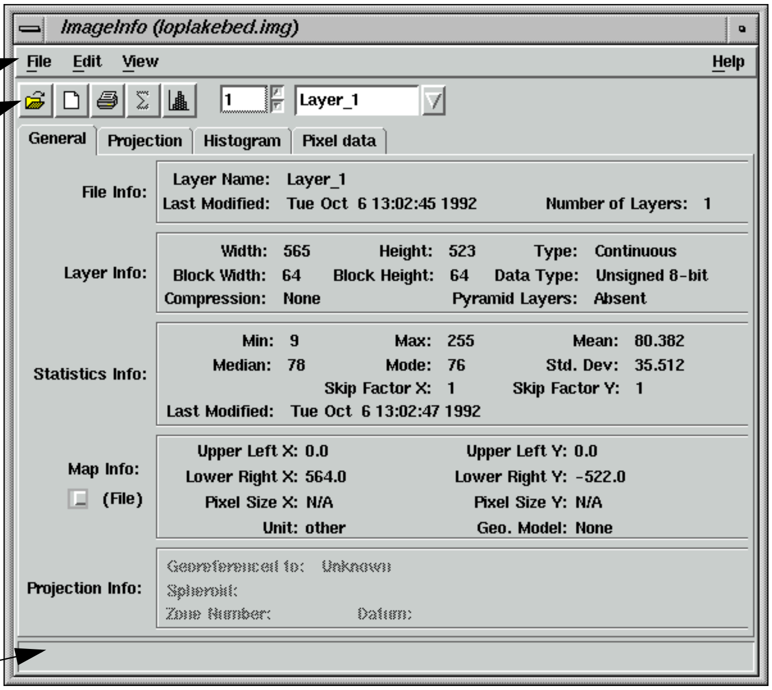
1. Select **Tools | Image Information** from the ERDAS IMAGINE menu bar.

   The ImageInfo dialog opens.

2. Select **File | Open** from the ImageInfo menu bar to select a file. You can also click on the Open icon in the ImageInfo tool bar to select a file.

3. In the Image Files dialog, click on **loplakebed.img** to select it and then click **OK**.

   The information for **loplakebed.img** displays in the ImageInfo dialog.



Menu bar

Tool bar

Status bar

ImageInfo (loplakebed.img)
File Edit View Help
1
Layer_1
General Projection Histogram Pixel data

File Info:
Layer Name: Layer_1
Last Modified: Tue Oct 6 13:02:45 1992
Number of Layers: 1

Layer Info:
Width: 565
Height: 523
Type: Continuous
Block Width: 64
Block Height: 64
Data Type: Unsigned 8-bit
Compression: None
Pyramid Layers: Absent

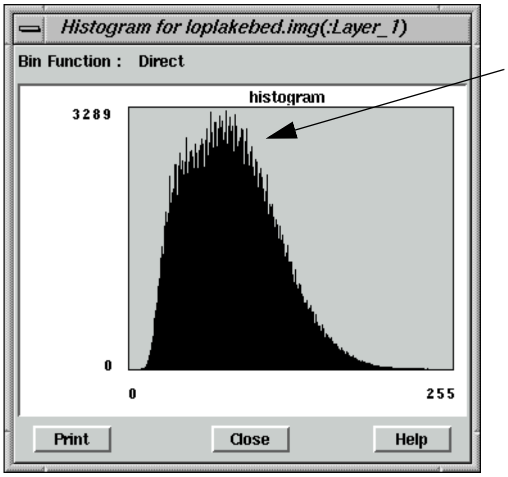
Statistics Info:
Min: 9
Max: 255
Mean: 80.382
Median: 78
Mode: 76
Std. Dev: 35.512
Skip Factor X: 1
Skip Factor Y: 1
Last Modified: Tue Oct 6 13:02:47 1992

Map Info: ☐ (File)
Upper Left X: 0.0
Upper Left Y: 0.0
Lower Right X: 564.0
Lower Right Y: -522.0
Pixel Size X: N/A
Pixel Size Y: N/A
Unit: other
Geo. Model: None

Projection Info:
Georeferenced to: Unknown
Spheroid:
Zone Number:
Datum:

4. In the ImageInfo dialog, select **View | Histogram**, or click the Histogram icon.

The histogram for **loplakebed.img** displays.



Bin Function : Direct

Histogram for loplakebed.img(:Layer_1)
histogram
3289
0
0
255
Print
Close
Help

The presence of spikes indicates the need for speckle reduction.

5. Select **File | New** from the ImageInfo dialog menu bar to open another ImageInfo dialog.

A second ImageInfo dialog opens.

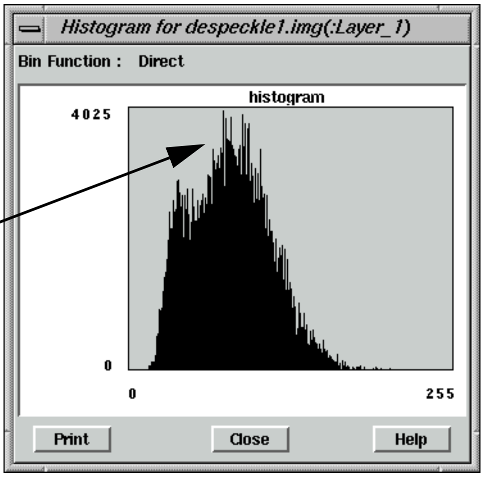
6. Click on the Open icon in the new ImageInfo dialog.

7. In the Open File dialog, select **despeckle1.img** from the directory in which you saved it and then click **OK**.

   The information for **despeckle1.img** displays in the ImageInfo dialog.

8. In the ImageInfo dialog, click the Histogram icon.

   The histogram for **despeckle1.img** displays.



After one pass, the spikes have been reduced. Also note the separation of two distinct classes in the data.

9. Repeat steps (5. through (8. of "Use Histograms to Evaluate Images" to view the subsequent passes of speckle reduction performed (**despeckle2.img**, **despeckle3.img**).

10. When finished, click **Close** in the Histogram viewers.

11. Select **File | Close** from the ImageInfo dialogs.

12. Select **File | Clear** in both Viewers.

Enhance Edges

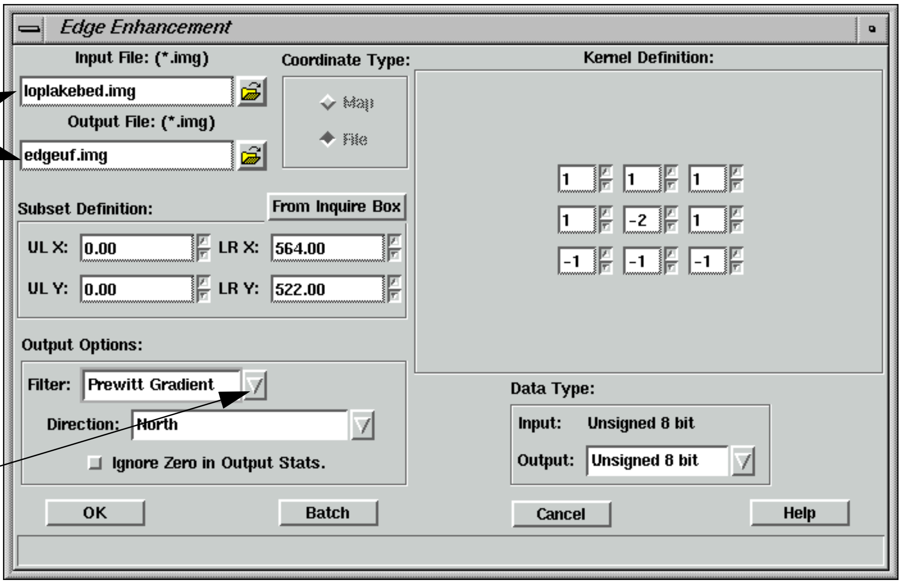
In this exercise, you create two images—one that is processed from the original image with the Edge Enhancement function, and one that is processed from the final result of the Speckle Suppression exercise. The objective is to demonstrate the effectiveness of Speckle Suppression prior to Edge Enhancement.

The Edge Enhancement functions in the IMAGINE Radar Interpreter module are similar to the Convolution and Neighborhood options in Image Interpreter.

NOTE: You can use the Edge Enhancement functions on any type of image—not just radar data.

1. From the **Radar Interpreter** menu, select **Edge Enhancement**.

   The Edge Enhancement dialog opens.



Select input and output files here

Select filter here

Input File: (*.img) loplakebed.img

Output File: (*.img) edgeuf.img

Coordinate Type: ◇ Map ◆ File

Kernel Definition:

1	1	1
1	-2	1
-1	-1	-1

Subset Definition: From Inquire Box

UL X: 0.00 LR X: 564.00

UL Y: 0.00 LR Y: 522.00

Output Options:

Filter: Prewitt Gradient

Direction: North

☐ Ignore Zero in Output Stats.

Data Type:

Input: Unsigned 8 bit

Output: Unsigned 8 bit

2. In the Edge Enhancement dialog under **Input File**, enter **loplakebed.img**.

3. Under **Output File**, enter **edgeuf.img** in the directory of your choice.

4. Under **Output Options**, click on the **Filter** popup list and select **Prewitt Gradient**.

5. Click **OK** in the Edge Enhancement dialog.

   The Edge Enhancement dialog closes and a Job Status dialog displays, indicating the progress of the function.

6. Repeat steps (1. through (5., using **despeckle3.img** as the **Input File** and **edgess.img** as the **Output File**.

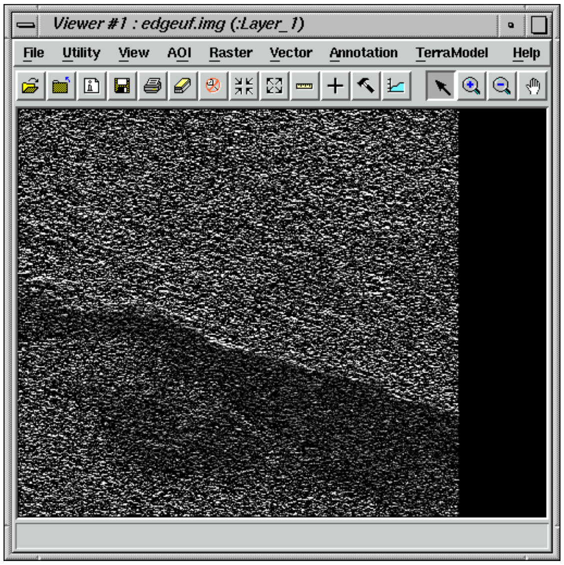
View Results

1. In Viewer #1, select **File | Open | Raster Layer**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, click on the file **edgeuf.img**, then click **OK**.

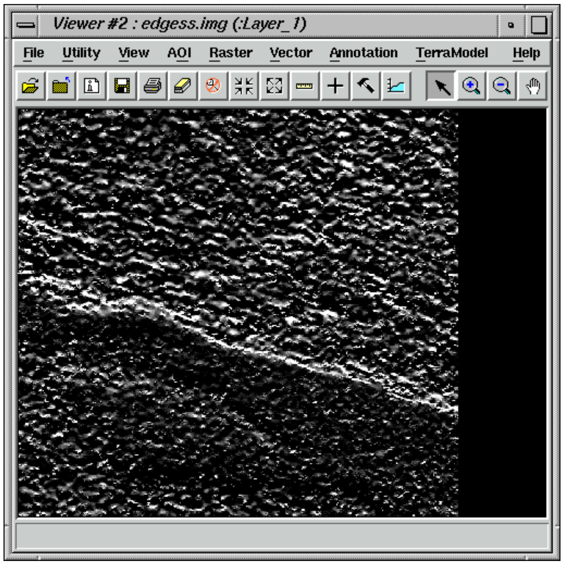
   This is the edge-filtered file derived from the unfiltered radar image file.


Viewer #1 : edgeuf.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

3. If necessary, start another Viewer. In Viewer #2, select **File | Open | Raster Layer**.

4. In the Select Layer To Add dialog, click the file **edgess.img**, then click **OK**.

   This is the edge filtered file derived from the speckle-suppressed file.


Viewer #2 : edgess.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

5. In the ERDAS IMAGINE menu bar, select **Session | Tile Viewers** to position and size the Viewers so that you can see both of them at once on the screen.

   The results should clearly show a more visible lake bed in the image that was speckle filtered (**edgess.img**). As an experiment, you may now want to take the unfiltered, edge-enhanced image (**edgeuf.img**) and pass it through the same Speckle Suppression process done previously. Comparing the result of this experiment with **edgess.img** should show whether it is better to perform speckle suppression before or after edge enhancement.

You can experiment with other edge enhancement filters or proceed to the next section.

6. When you are finished comparing the images, select **File | Clear** in Viewer #1 and Viewer #2.

Enhance Image

The IMAGINE Radar Interpreter module provides three image enhancement categories:

- Wallis Adaptive Filter
- luminance modification
- sensor merge

Wallis Adaptive Filter

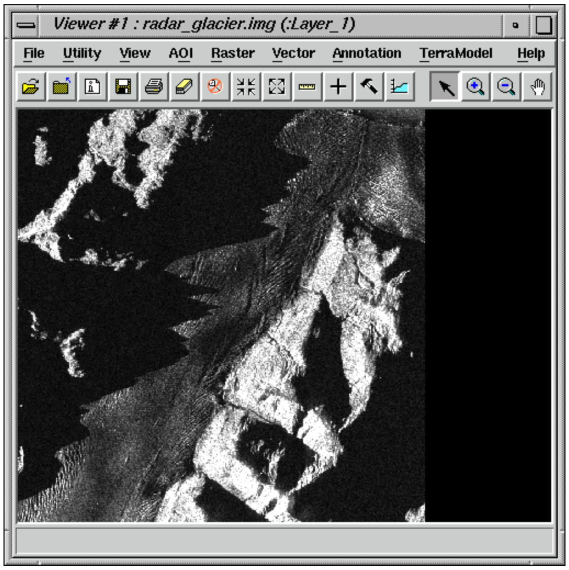
The Wallis adaptive filter is designed to adjust the contrast stretch of an image using only the values within a local region (defined by the window size), which makes it widely applicable. Three possible implementations of this technique are provided: Bandwise, IHS, and PC.

- In the Bandwise operation, the adaptive filter is passed over each band sequentially.
- In the IHS implementation, the input RGB image is transformed into IHS space. The adaptive filter is only passed over the intensity (I) component. The image is then transformed back into RGB.
- In the PC implementation, the input bands are transformed into principal components. The filter is only passed over PC-1. An inverse principal component transform is then performed.

In this section, you apply the Wallis adaptive filter function to an image and observe the results.

Wallis Adaptive Filter

Make sure the IMAGINE Radar Interpreter module is running, and display the file **radar_glacier.img** in a Viewer.


Viewer #1 : radar_glacier.img (:Layer_1)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

1. In the **Radar Interpreter** menu, select **Speckle Suppression**.

   The Radar Speckle Suppression dialog opens.

2. In the Radar Speckle Suppression dialog, enter **radar_glacier.img** as the **Input File**.
3. Type in **despeckle4.img** (in the directory of your choice) as the **Output File**.
4. Select **Gamma-MAP** from the **Filter** popup list.
5. Click **OK** in the Radar Speckle Suppression dialog to filter the image.

   The Radar Speckle Suppression dialog closes and a Job Status dialog displays, indicating the progress of the function.

6. Click **OK** in the Job Status dialog when the process is complete.
7. Select **Image Enhancement** from the **Radar Interpreter** menu.

   The **Image Enhancement** menu opens.

Click Wallis Adaptive Filter

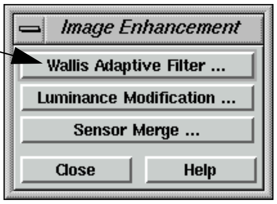
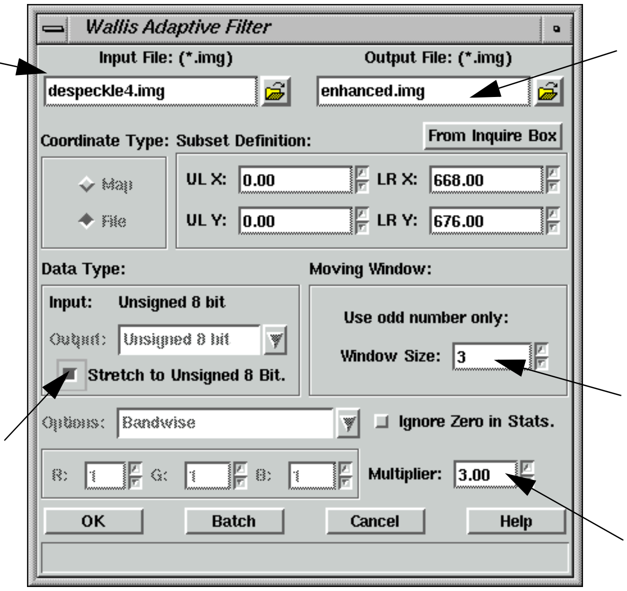


Image Enhancement
Wallis Adaptive Filter ...
Luminance Modification ...
Sensor Merge ...
Close
Help

8. Click on **Wallis Adaptive Filter** in the **Image Enhancement** menu.

   The Wallis Adaptive Filter dialog opens.



Enter the name of the input file here

Enter the name of the output file here

Set window size here

Click this check box to activate this option

Enter contrast multiplier here

Wallis Adaptive Filter

Input File: (*.img): despeckle4.img

Output File: (*.img): enhanced.img

Coordinate Type: ○ Map ● File

Subset Definition: UL X: 0.00 LR X: 668.00 UL Y: 0.00 LR Y: 676.00

Data Type: Input: Unsigned 8 bit

Output: Unsigned 8 bit

☑ Stretch to Unsigned 8 Bit.

Moving Window: Use odd number only: Window Size: 3

Options: Bandwise

☐ Ignore Zero in Stats.

R: 1 G: 1 B: 1

Multiplier: 3.00

9. In the Wallis Adaptive Filter dialog under **Input File**, enter the file **despeckle4.img**.

10. Under **Output File**, enter the name **enhanced.img** in the directory of your choice.

11. Under **Data Type**, click **Stretch to Unsigned 8 Bit**.

12. Under **Moving Window**, confirm that the **Window Size** is set to **3**.

    Rough images usually require smaller window sizes (3 × 3), whereas smooth, or cleaner, images can tolerate larger window sizes.

13. Set the **Multiplier** to **3.00**.

14. Click **OK** in the Wallis Adaptive Filter dialog.

    The Wallis Adaptive Filter dialog closes and a Job Status dialog displays, indicating the progress of the function.

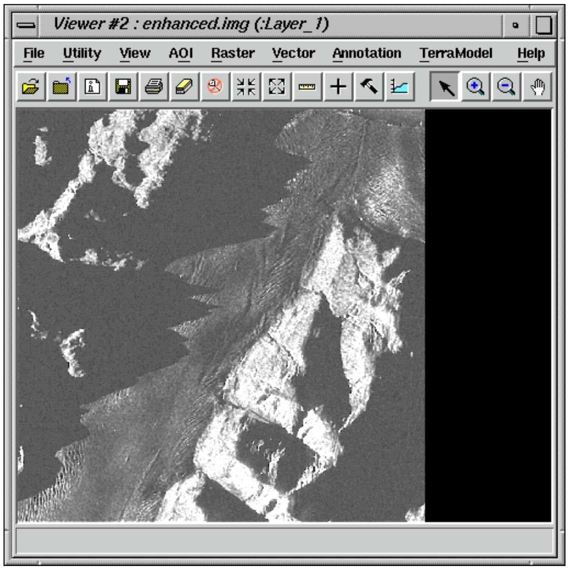
15. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of Viewer #2, select **File | Open | Raster Layer**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select the file **enhanced.img** and then click **OK**.


Viewer #2 : enhanced.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

3. Examine the differences between the two files.

4. When you are finished comparing the images, select **File | Clear** in Viewer #1 and Viewer #2.

Apply Sensor Merge

Next, you apply the **Sensor Merge** function to an image and observe the results.

This package of algorithms enables you to combine imagery from different sensors. Examples of this would be radar with TM imagery or multifrequency radar with aeromagnetic data. Three different families of techniques are available: Principal Component, IHS, and Multiplicative (these are similar to those in the Wallis Adaptive Filter option).

Principal Component

In using the **Principal Component** techniques, you have the option to modify the gray scale image in any of the following ways.

- **Remap**—rescales the gray scale image to the range of PC-1.
- **Hist. Match**—matches the histogram of the gray scale image to PC-1.
- **Multiply—**rescales the gray scale image into the 0-1 range and then multiplies the gray scale by PC-1.
- **None**—replaces PC-1 with the input gray scale image.

IHS

Using the **IHS** family, two options exist.

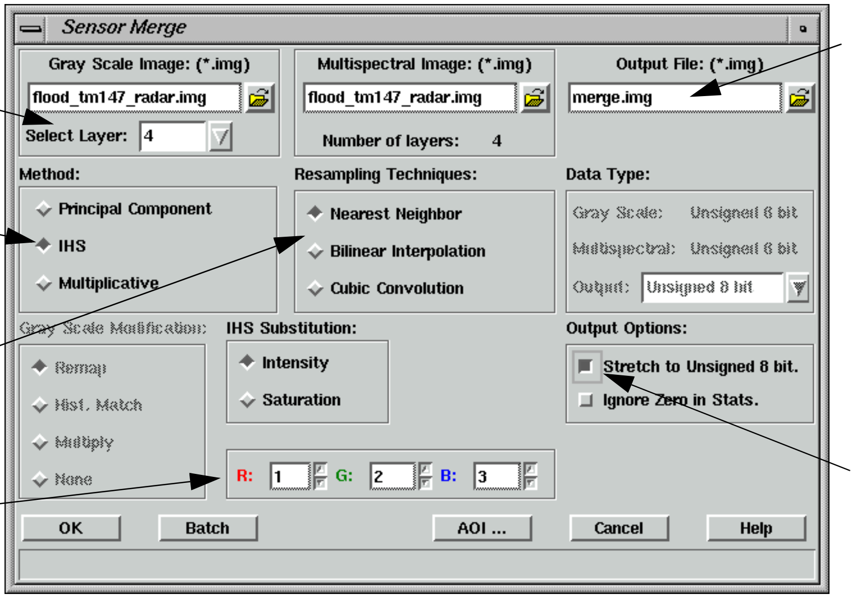
- **Intensity**—rescales the gray scale image to the numerical range of the intensity (I) and then substitutes it for I.
- **Saturation**—rescales the gray scale image to the numerical range of saturation (S) and then substitutes it for S.

Multiplicative

The **Multiplicative** technique remaps the gray scale image to a 0-1 range. Each band is then sequentially multiplied by the remapped gray scale image.

1. If it is not already open, open the **Image Enhancement** menu by selecting **Image Enhancement** from the **Radar Interpreter** menu.
2. In the **Image Enhancement** menu, select **Sensor Merge**.

   The Sensor Merge dialog opens.



Sensor Merge

Gray Scale Image: (*.img) flood_tm147_radar.img

Select Layer: 4

Multispectral Image: (*.img) flood_tm147_radar.img

Number of layers: 4

Output File: (*.img) merge.img

Method:
- ○ Principal Component
- ● IHS
- ○ Multiplicative

Resampling Techniques:
- ● Nearest Neighbor
- ○ Bilinear Interpolation
- ○ Cubic Convolution

Data Type:
- Gray Scale: Unsigned 8 bit
- Multispectral: Unsigned 8 bit
- Output: Unsigned 8 bit

Gray Scale Modification:
- ● Remap
- ○ Hist. Match
- ○ Multiply
- ○ None

IHS Substitution:
- ● Intensity
- ○ Saturation

Output Options:
- ☑ Stretch to Unsigned 8 bit.
- ☐ Ignore Zero in Stats.

R: 1 G: 2 B: 3

Click on this popup list to select the radar image layer

Select IHS for the method

Select Nearest Neighbor as the resampling technique

Enter the appropriate layers to colors here

Enter the name of the output file here

Click this checkbox to activate this option

3. In the Sensor Merge dialog under **Gray Scale Image**, enter **flood_tm147_radar.img**.
4. Click on the **Select Layer** popup list and select **4** (the radar image layer).
5. Enter **flood_tm147_radar.img** under **Multispectral Image**.
6. Enter **merge.img** as the **Output File** (in the directory of your choice).
7. Under **Method**, click **IHS**.
8. Under **Resampling Techniques**, click **Nearest Neighbor**.
9. Make sure that **Intensity** is selected under **IHS Substitution**.
10. In the **R**, **G**, and **B** boxes, enter **1** for **R**, **2** for **G**, and **3** for **B** (the TM image layers).
11. Under **Output Options**, click **Stretch to Unsigned 8 bit**.
12. Click **OK** in the Sensor Merge dialog.

    The Sensor Merge dialog closes and a Job Status dialog displays, indicating the progress of the function.

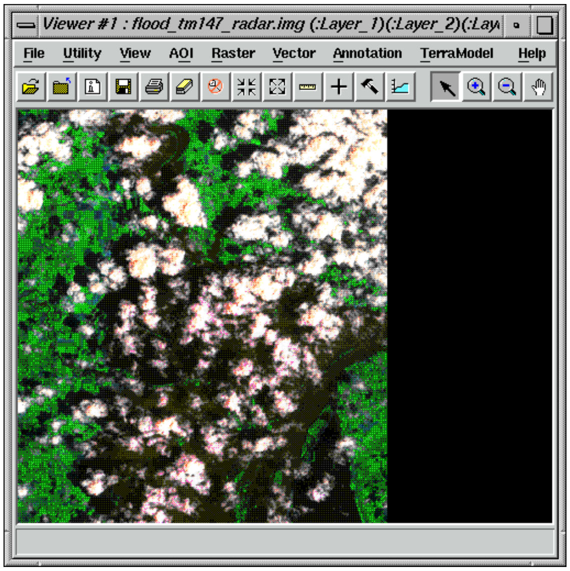
13. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of Viewer #1, select **File | Open | Raster Layer**.

   The Select Layer To Add dialog opens.

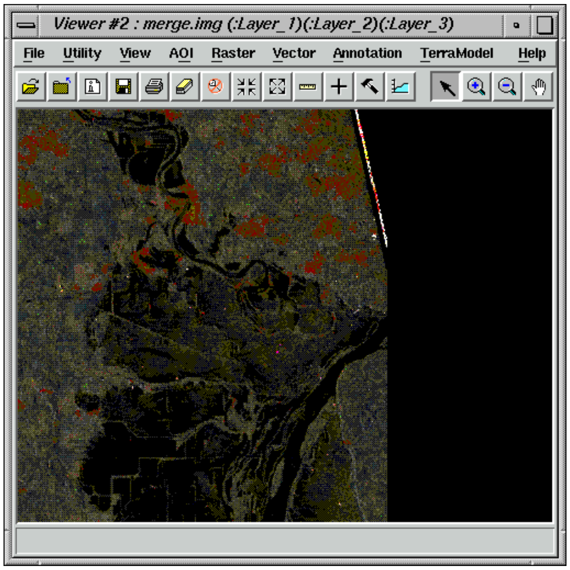
2. In the Select Layer To Add dialog, click on the file **flood_tm147_radar.img**.
3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
4. Under **Layers to Colors**, select **1** for **Red**, **2** for **Green**, and **3** for **Blue**.
5. Click **OK** in the Select Layer To Add dialog.


Viewer #1 : flood_tm147_radar.img (:Layer_1)(:Layer_2)(:Lay
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

6. In Viewer #2, select **File | Open | Raster Layer**.

   The Select Layer To Add dialog opens.

7. In the Select Layer To Add dialog, click the file **merge.img**.
8. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
9. Under **Layers to Colors**, select **1** for **Red**, **2** for **Green**, and **3** for **Blue**.
10. Click **OK**.


Viewer #2 : merge.img (:Layer_1)(:Layer_2)(:Layer_3)
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

11. Examine the difference between the two files.

12. When you are finished comparing the images, select **File | Clear** in Viewer #1 and Viewer #2.

13. Click **Close** in the **Image Enhancement** menu.

Apply Texture Analysis

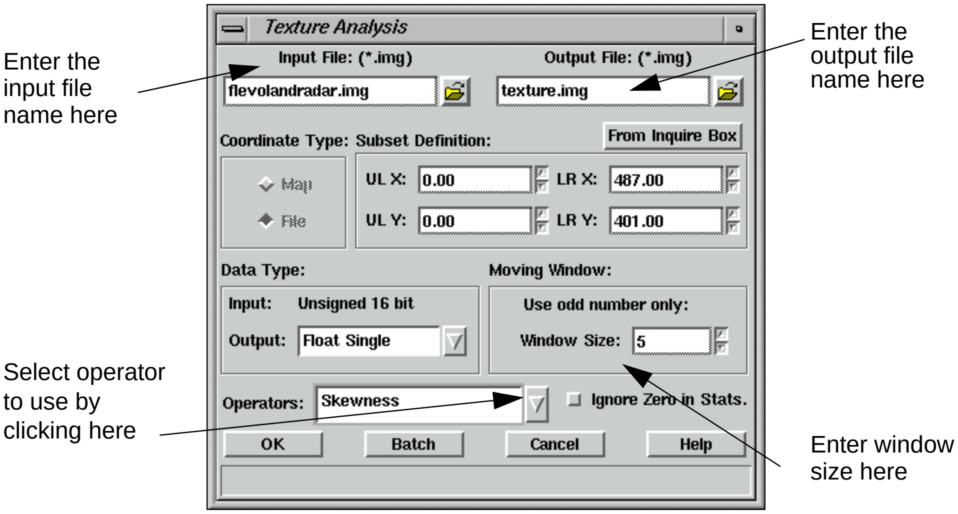
Next, apply the **Texture Analysis** function to an image and observe the results.

The radar data's sensitivity to texture is an advantage over other types of imagery where texture is not a quantitative characteristic.

NOTE: Texture analysis has been shown to be useful for geologic discrimination and vegetation classification.

1. From the **Radar Interpreter** menu, select **Texture Analysis**.

The Texture Analysis dialog opens.



Input File: (*.img) flevolandradar.img

Output File: (*.img) texture.img

Coordinate Type: ○ Map ● File

Subset Definition:

UL X: 0.00
LR X: 487.00
UL Y: 0.00
LR Y: 401.00

Data Type:

Input: Unsigned 16 bit
Output: Float Single

Moving Window:

Use odd number only:
Window Size: 5

Operators: Skewness

☐ Ignore Zero in Stats.

Enter the input file name here

Enter the output file name here

Select operator to use by clicking here

Enter window size here

2. In the Texture Analysis dialog, enter **flevolandradar.img** as the **Input File**.

3. Enter **texture.img** (in the directory of your choice) as the **Output File**.

4. Click on the **Operators** popup list and select **Skewness**.

5. Under **Moving Window**, enter a **Window Size** of **5**.

6. Click **OK** in the Texture Analysis dialog.

   The Texture Analysis dialog closes and a Job Status dialog displays, indicating the progress of the function.

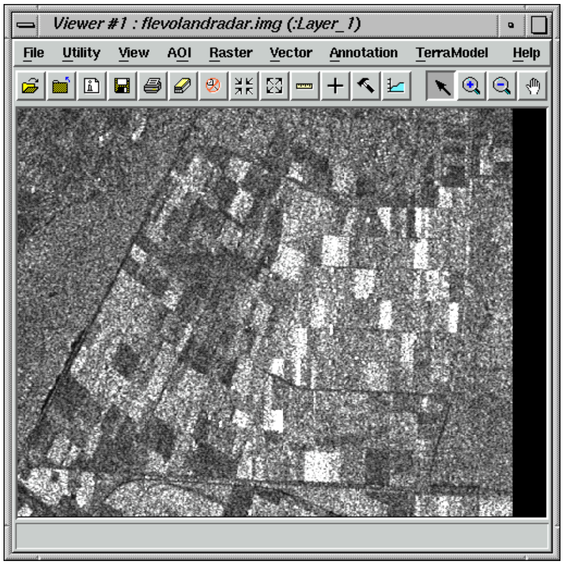
7. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of Viewer #1, select **File | Open | Raster Layer**.

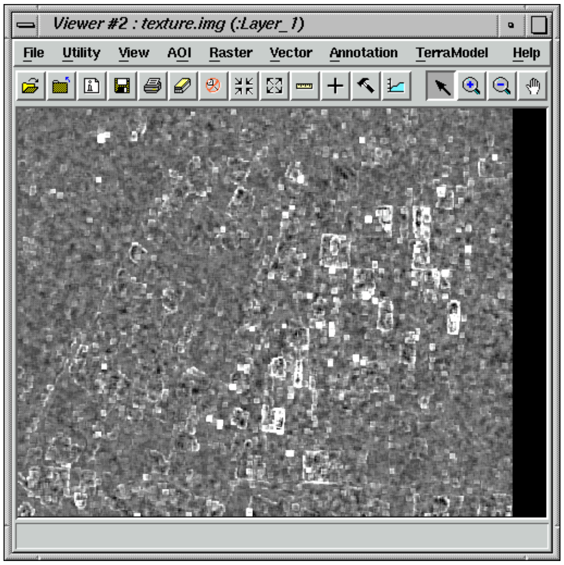
2. In the Select Layer To Add dialog, click on the file **flevolandradar.img.** This is an agricultural subscene from Flevoland, Holland. This image is from the ERS-1 satellite in C-band with 20-meter pixels.

3. Click **OK** in the Select Layer To Add dialog.


Viewer #1 : flevolandradar.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

4. In Viewer #2, select **File | Open | Raster Layer**.

5. In the Select Layer To Add dialog, click the file **texture.img**. then click **OK**.


Viewer #2 : texture.img (:Layer_1)
File Utility View AOI Raster Vector Annotation TerraModel Help

6. Examine the difference between the two files.

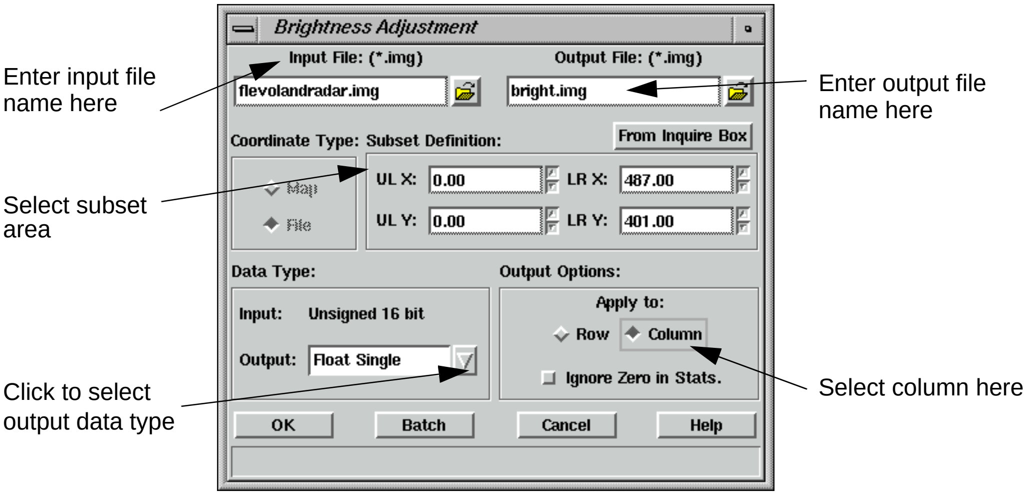
7. When you are finished comparing the images, select **File | Clear** in Viewer #1 and Viewer #2.

Adjust Brightness

The **Brightness Adjustment** function works by adjusting pixel DN values so that each line of constant range has the same average. In this way, the image is adjusted to have an overall, even brightness. Therefore, you must tell ERDAS IMAGINE whether the lines of constant range are stored in rows or columns. This depends on the flight path of the sensor and the output raster it produces.

1. Select **Adjust Brightness** from the **Radar Interpreter** menu.

   The Brightness Adjustment dialog opens.


Brightness Adjustment
Enter input file name here
Enter output file name here
Select subset area
Click to select output data type
Select column here
From Inquire Box
OK
Batch
Cancel
Help

Input File: (*.img) flevolandradar.img

Output File: (*.img) bright.img

Coordinate Type: ○ Map ● File

Subset Definition: UL X: 0.00 LR X: 487.00 UL Y: 0.00 LR Y: 401.00

Data Type: Input: Unsigned 16 bit Output: Float Single

Output Options: Apply to: ○ Row ● Column

☐ Ignore Zero in Stats.

2. In the Adjust Brightness dialog under **Input File**, enter the name of the input file, **flevolandradar.img**.

3. Under **Output File**, enter the name of the output file, **bright.img**, in the directory of your choice.

4. Under **Subset Definition**, select a subset of the file if you want to apply the function to a portion of the image rather than the entire image.

5. Select the **Data Type** under **Output** file. The default is **Float Single**, which is recommended to save disk space.

6. Under **Apply to** in the **Output Options**, select **Column**.

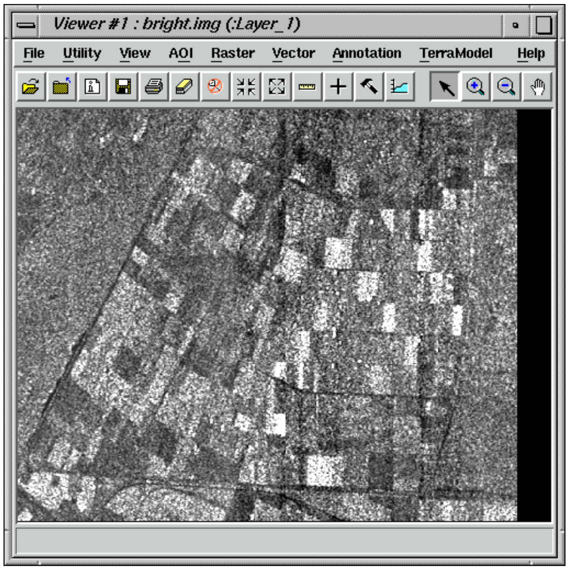
   You can often tell whether the data are stored in rows or columns by looking at the image header data or by consulting documentation supplied with the data.

   Use the **Data View** *option from the Import/Export dialog or* **Tools | View Binary Data** *from the ERDAS IMAGINE menu bar to read the image header data.*

7. Click **OK** in the Adjust Brightness dialog.

The Adjust Brightness dialog closes and a Job Status dialog displays, indicating the progress of the function.

8. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).
9. Select **File | Open | Raster layer** in the Viewer menu bar.
10. Navigate to the appropriate directory, then select **bright.img**.



11. After processing is completed, you must view and evaluate the resultant image and decide if another pass is needed to obtain the results you want.

See "Chapter 5: Enhancement" *in the* ERDAS Field Guide *for theoretical information.*

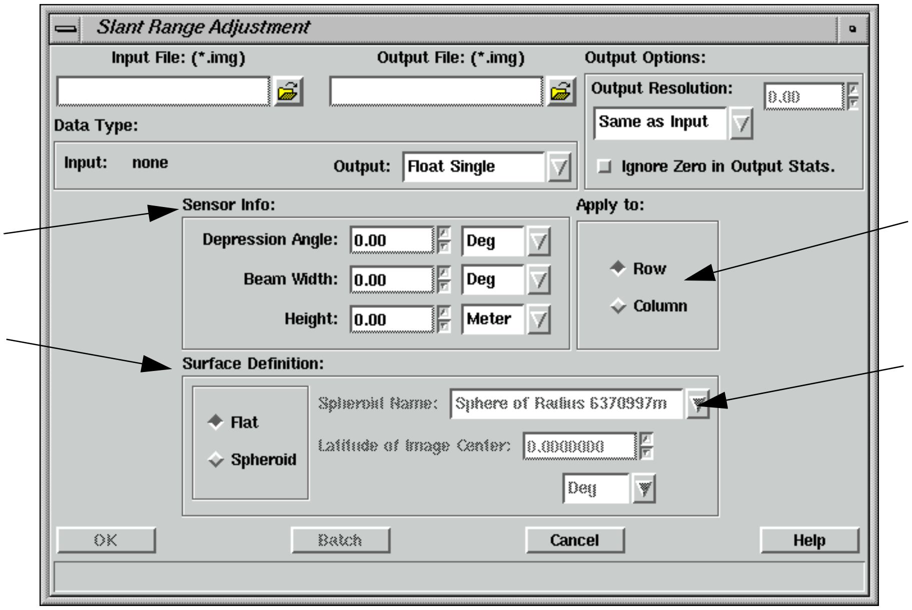
Adjust Slant Range

This section does not take you through an actual demonstration of the **Slant Range Adjustment** function, since the full image is required. However, when using this function, you follow the next series of steps.

The Slant Range Adjustment function applies only to radar data.

1. Select **Adjust Slant Range** from the **Radar Interpreter** menu.

The Slant Range Adjustment dialog opens.



Output Resolution: 0.00

Same as Input

☐ Ignore Zero in Output Stats.

Input: none

Output: Float Single

Depression Angle: 0.00 Deg

Beam Width: 0.00 Deg

Height: 0.00 Meter

● Row

● Flat

Spheroid Name: Sphere of Radius 6370997m

Latitude of Image Center: 0.0000000 Deg

Enter sensor information here

Indicate Surface Definition here

Select row or column here

Click here for a list of spheroid types

2. In the Slant Range Adjustment dialog under **Input File**, enter the name of the input file.

3. Under **Output File**, enter the name of the output file in the directory of your choice.

4. Under **Data Type**, select the data type for the **Output File** by clicking on the popup menu. The default is **Float Single**, which is recommended to save disk space but still retain precision.

5. Under **Sensor Info**, you must enter sensor-specific information that is obtained either from the data header information or from the data distributor.

Use the **Data View** *option from the Import/Export dialog or* **Tools | View Binary Data** *from the ERDAS IMAGINE menu bar to read the image header data.*

6. Under **Apply to**, select **Row** or **Column**. See the previous section on "Adjust Brightness" for information about row and column selection.

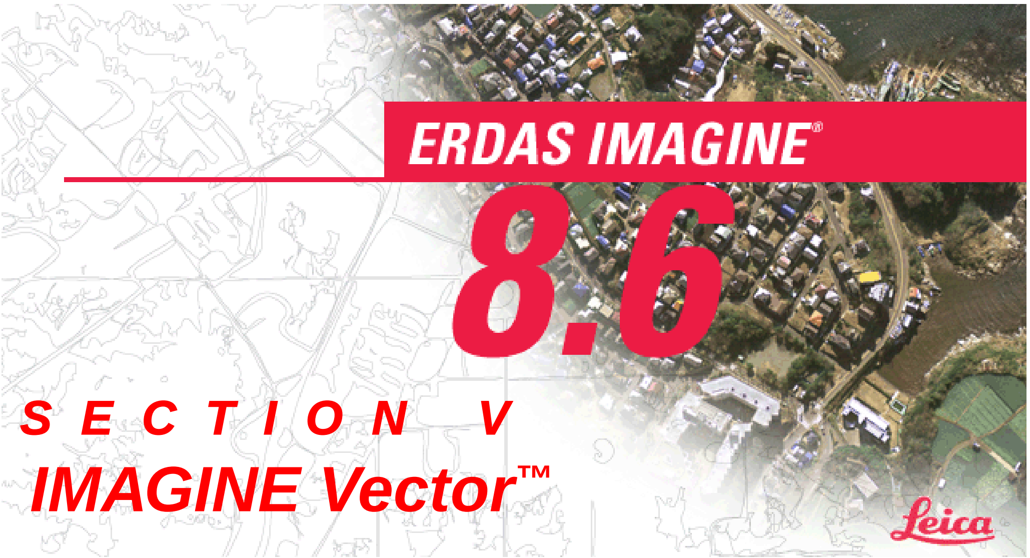
7. Under the **Surface Definition** section:
   - select **Flat** for shuttle or aircraft data, such as SIR-A or B, or AIRSAR, or
   - select **Spheroid** for satellite data (ERS-1, Fuyo (JERS-1), RADARSAT, etc.)

8. Click **OK** in the Slant Range Adjustment dialog.

   A Job Status dialog displays, indicating the progress of the function.

9. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

10. After processing is completed, you must view and evaluate the resultant image and decide if another pass is needed to obtain the desired results.


ERDAS IMAGINE®
8.6
Leica

SECTION V

IMAGINE Vector™

CHAPTER TWENTY ONE

IMAGINE Vector™

Introduction

The IMAGINE Vector capabilities are designed to provide you with an integrated GIS package for raster and vector processing. The vector tools in ERDAS IMAGINE are based on the ESRI vector data model, therefore ArcInfo vector coverages, ESRI shapefiles, and ESRI SDE vectors can be used in ERDAS IMAGINE with no conversion.

By integrating raster and vector data into one system, you can compile a complete data base of your study area. You can overlay vectors onto current and accurate raster layers to update your vector information, including attributes. You can also use vectors to define an area of interest for an operation, such as classification or enhancement.

The IMAGINE Vector capabilities are divided into these levels:

- **Native**—vector capabilities that are native to ERDAS IMAGINE. These functions provide vector and attribute display and query using multiple selection tools, as well as vector creation and editing.
- **IMAGINE Vector module**—an add-on module for ERDAS IMAGINE that provides you with additional vector utility tools and importers/exporters of various vector formats. The utility tools include coverage clean, build, transform, create-label, raster to vector and vector to raster conversion, and a table tool for an INFO database. The importers/exporters handle the following data formats: DFAD, DGN, DLG, DXF, ETAK, IGES, SDTS, TIGER, and VPF.

This tour guide mainly discusses the IMAGINE Vector module. Depending on the package you are using, you may not be able to perform all of the steps. The **Native** *capabilities are discussed in* "Chapter 4: Vector Querying and Editing" *on page 95.*

This tour guide covers the following topics:

- copying vector data
- manipulating information attributes
- displaying vector layers
- changing vector symbology
- building and cleaning layers

The data used in this tour guide are in the <IMAGINE_HOME>/examples directory. Replace <IMAGINE_HOME> with the directory where ERDAS IMAGINE is installed on your system (e.g., /usr/imagine/850).

A Digitizing Template is supplied in the information packet for ERDAS IMAGINE V8.6.

The vector data provided with ERDAS IMAGINE do not have write permission, therefore you need to copy them to another directory so that you can edit them. However, vector layers are not simple files. They are stored in directories called workspaces. Using the simple UNIX copy commands does not copy all of the necessary files. You must use the **Copy** utility provided with ERDAS IMAGINE.

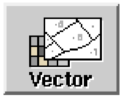
NOTE: If you do not have the full IMAGINE Vector module, you do not need to move the data, since you cannot edit it.

Approximate completion time for this tour guide is 55 minutes.

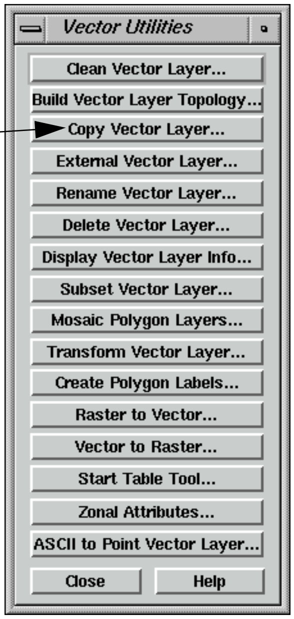
Copy Vector Data

Move to the directory where you want to create your workspace. Start ERDAS IMAGINE from this directory. Make sure this is a directory in which you have read/write permissions.

1. Click on the Vector icon from the ERDAS IMAGINE icon panel.



The **Vector Utilities** menu opens.


Vector Utilities
Clean Vector Layer...
Build Vector Layer Topology...
Copy Vector Layer...
External Vector Layer...
Rename Vector Layer...
Delete Vector Layer...
Display Vector Layer Info...
Subset Vector Layer...
Mosaic Polygon Layers...
Transform Vector Layer...
Create Polygon Labels...
Raster to Vector...
Vector to Raster...
Start Table Tool...
Zonal Attributes...
ASCII to Point Vector Layer...
Close
Help

Click here to copy sample vector layer

Depending on the package you have, you may not see all of these utilities.

IMAGINE Vector

This menu lists most of the tools of the IMAGINE Vector module. Through simple dialogs generated from this menu, you can:

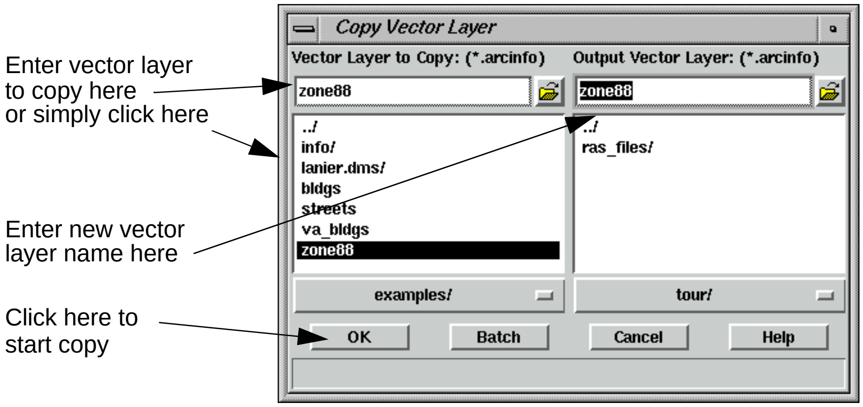
- **Clean** a vector coverage
- **Build** topology
- **Copy, Rename**, **Delete**, and **Externalize** vector layers (native functions)
- **Subset** vector layers
- **Mosaic** polygon layers
- **Transform** vector coverages from digitizer units to real-world units
- Create polygon **Labels**
- Convert **Raster to Vector** layers and **Vector to Raster** layers
- Manipulate information tables using the **Table Tool**
- Generate polygon attributes from a background image (**Zonal Attributes**)
- Convert an **ASCII File to a Point Layer**

NOTE: Creating vector layers and vector editing tools are available through the Viewer. If you have the IMAGINE Vector module, you can also use the Import icon to access various external vector data types, such as DFAD, DGN, DLG, DXF, SDTS, TIGER, and VPF.

The vector utilities in the **Vector Utilities** *menu should not be run on open vector layers. Close the layer you are using before running the utility, and do not attempt to open the layer until the process is complete.*

2. Select **Copy Vector Layer** from the **Vector Utilities** menu.

The Copy Vector Layer dialog opens.



Enter vector layer to copy here or simply click here

Enter new vector layer name here

Click here to start copy

Vector Layer to Copy: (*.arcinfo) zone88

Output Vector Layer: (*.arcinfo) zone88

3. In the Copy Vector Layer dialog under **Vector Layer to Copy**, enter the file name **zone88**.
4. Under **Output Vector Layer**, enter **zone88** in the directory of your choice.
5. Click **OK** in the Copy Vector Layer dialog.

   A Job Status dialog displays to track the progress of the function. When copying is complete, you are ready to proceed with this tour guide.

6. Click **OK** in the Job Status dialog (if it does not dismiss automatically).

   The Copy Vector Layer dialog automatically closes.

7. Click **Close** in the **Vector Utilities** menu to dismiss it.

Manipulate Info Files

The Table Tool is a utility for managing INFO files. It allows you to view, edit, relate, import/export, copy, rename, delete, merge, and create INFO tables.

Prepare

ERDAS IMAGINE must be running and you must have completed the previous section, "Copy Vector Data" on page 628.

Start Table Tool

1. Click the Vector icon in the ERDAS IMAGINE icon panel.

   The **Vector Utilities** menu opens.

2. In the **Vector Utilities** menu, click on **Start Table Tool**.

The Table Tool opens.

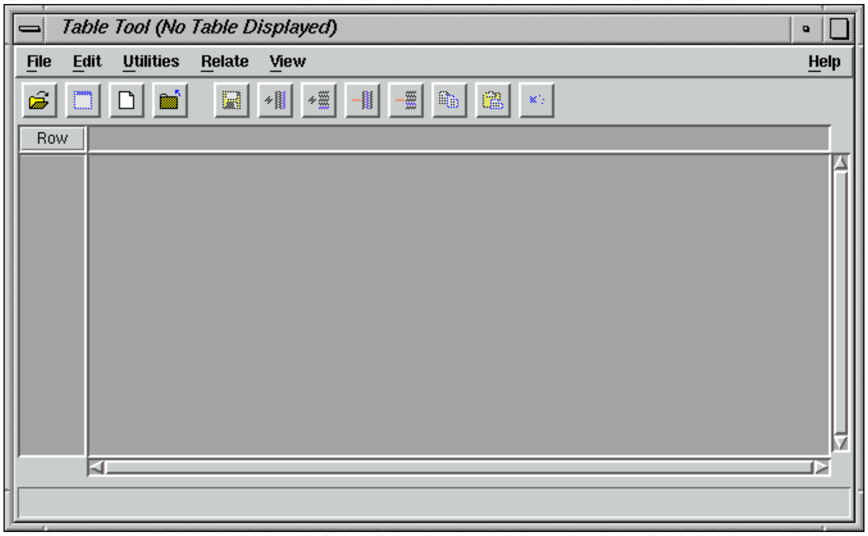


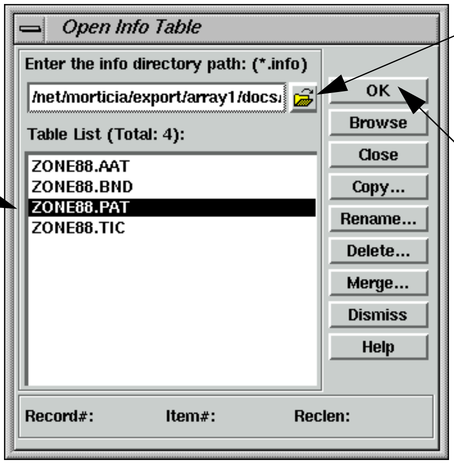
Table Tool (No Table Displayed)
File
Edit
Utilities
Relate
View
Help
Row

3. Click **Close** in the **Vector Utilities** menu to clear it from the screen.

Display an INFO File

4. In the Table Tool, click the Open icon or select **File | Open** from the menu bar.

The Open Info Table dialog displays.


Open Info Table
Enter the info directory path: (*.info)
Table List (Total: 4):
OK
Browse
Close
Copy...
Rename...
Delete...
Merge...
Dismiss
Help
Record#:
Item#:
Reclen:

/net/morticia/export/array1/docs/

ZONE88.AAT
ZONE88.BND
ZONE88.PAT
ZONE88.TIC

Click here to select the info directory path

Click here to select ZONE88.PAT

Click here to open the selected info table

5. In the Open Info Table dialog, click the Open icon to select the **info directory path**.

The Enter the info directory path dialog opens.

6. Under **Directory**, check to be sure that the directory to which you copied **ZONE88** is listed. If it is not, type that directory path in under **Filename** and press Return on your keyboard.

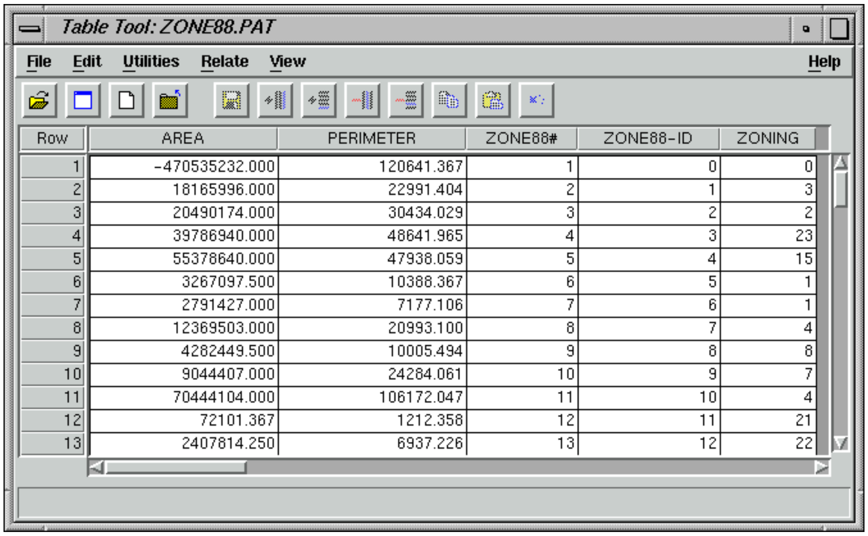
7. In the file list under **Filename**, click **info** and then click **OK**.

   The directory path and table list for **info** display in the Open Info Table dialog.

   NOTE: You can double-click on any table name to browse the table contents before clicking **OK** *to open that table.*

8. In the Open Info Table dialog under **Table List**, click **ZONE88.PAT** and then click **OK**.

   The information for **ZONE88.PAT** displays in the Table Tool CellArray.



Row	AREA	PERIMETER	ZONE88#	ZONE88-ID	ZONING
1	-470535232.000	120641.367	1	0	0
2	18165996.000	22991.404	2	1	3
3	20490174.000	30434.029	3	2	2
4	39786940.000	48641.965	4	3	23
5	55378640.000	47938.059	5	4	15
6	3267097.500	10388.367	6	5	1
7	2791427.000	7177.106	7	6	1
8	12369503.000	20993.100	8	7	4
9	4282449.500	10005.494	9	8	8
10	9044407.000	24284.061	10	9	7
11	70444104.000	106172.047	11	10	4
12	72101.367	1212.358	12	11	21
13	2407814.250	6937.226	13	12	22

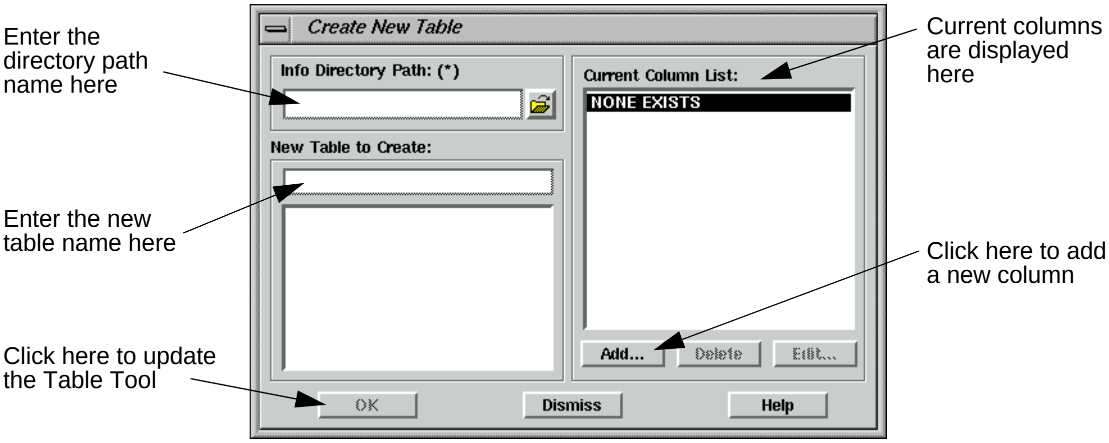
Create a New Table

1. In the Table Tool, click on the New icon.

   A second Table Tool displays. The new Table Tool CellArray is blank.

2. In the new Table Tool, select **File | New**.

   The Create New Table dialog opens.


Create New Table
Info Directory Path: (*)
New Table to Create:
Current Column List:
NONE EXISTS
Add...
Delete
Edit...
OK
Dismiss
Help

Enter the directory path name here

Enter the new table name here

Click here to update the Table Tool

Current columns are displayed here

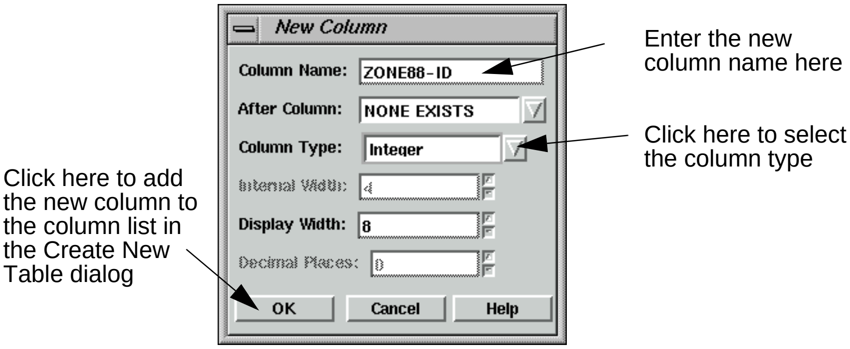
Click here to add a new column

3. In the Create New Table dialog, enter the name of an **Info Directory Path** in which you have write permission, or click on the Open icon to select the directory to put the new table in.

   The table list for the path you selected displays in the Create New Table dialog.

4. In the Create New Table dialog under **New Table to Create**, enter the name **ZONE88_NEW.PAT**, and then click **Add**.

   The New Column dialog opens.


New Column
OK
Cancel
Help

Column Name: ZONE88-ID
After Column: NONE EXISTS
Column Type: Integer
Internal Width: 4
Display Width: 8
Decimal Places: 0

Enter the new column name here

Click here to select the column type

Click here to add the new column to the column list in the Create New Table dialog

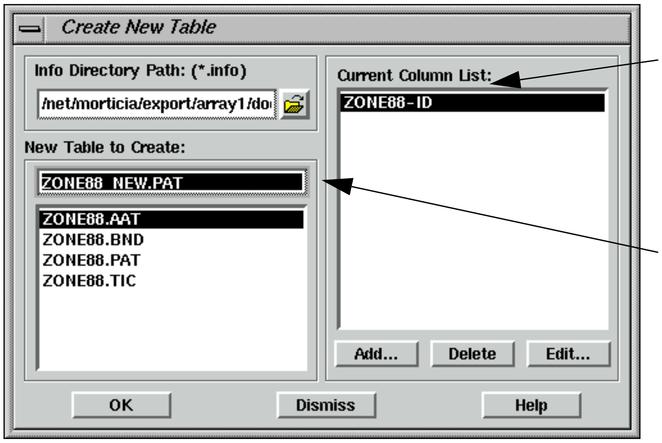
5. In the New Column dialog next to **Column Name** enter **ZONE88-ID**.

6. Next to **Column Type** click on the popup list to select **Integer**.

7. Click **OK** in the New Column dialog.

8. Click **OK** in the Create New Table dialog.

   The new table is entitled **ZONE88_NEW.PAT** and has a column entitled **ZONE88-ID**.


Create New Table
Info Directory Path: (*.info)
/net/morticia/export/array1/do
New Table to Create:
ZONE88_NEW.PAT
ZONE88.AAT
ZONE88.BND
ZONE88.PAT
ZONE88.TIC
Current Column List:
ZONE88-ID
Add...
Delete
Edit...
OK
Dismiss
Help

Column associated with new table

New table name

NOTE: If you create a new table by clicking **File | New** *in an existing table, the existing table is used as a template and the column definitions of the existing table are listed in the Create New Table dialog as the default columns of the new table.*

Copy From One Table to Another

1. In the **ZONE88.PAT** Table Tool, click in the column title of **ZONE88-ID** to select that column.

   The column is highlighted in blue.

2. In the **ZONE88.PAT** Table Tool, right-hold in the column title **ZONE88-ID** to select **Column Options | Copy**.

3. In the **ZONE88_NEW.PAT** Table Tool, click on the column title **ZONE88-ID** to select it and then right-hold **Column Options | Paste**.

   The ID numbers for **ZONE88-ID** are entered into the new table, and the **ZONE88-ID** column is highlighted in blue.

Add a Column to the Table

1. In the **ZONE88_NEW.PAT** Table Tool, select **Edit | Add A Column** from the menu bar.

   The Add Column dialog opens. This dialog is similar to the New Column dialog.

2. In the Add Column dialog next to **Column Name** enter **NEW_ZONING**.

3. Next to **Column Type** click on the popup list to select **Integer**.

4. Click **OK** in the Add Column dialog.

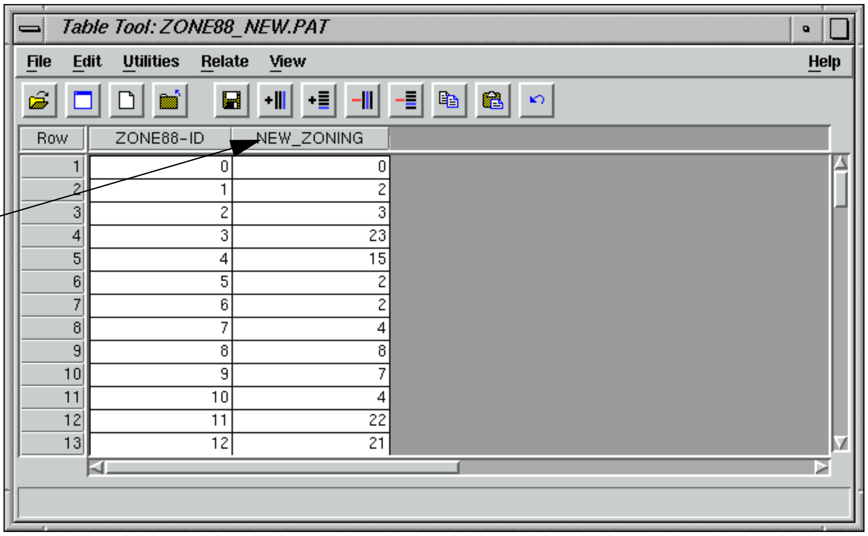
   The **NEW_ZONING** column is added to **ZONE88_NEW.PAT**.

Relate the Tables

Let’s say that you want to compare the **Zoning** information in **ZONE88.PAT** with some new zoning information that you have just received. First, you would need to enter the new zoning information in the **NEW_ZONING** column of the **ZONE88_NEW.PAT** Table Tool.

1. Follow the example and enter imaginary numbers, which somewhat differ from the **ZONING** numbers, into the **NEW_ZONING** column. (Enter the numbers by clicking in each cell, typing the number, and pressing Return on your keyboard.)

   NOTE: For this example, it would suffice to only enter numbers for rows 1 through 13. To enter numbers, select **Edit | Enable Editing**.



Row	ZONE88-ID	NEW_ZONING
1	0	0
2	1	2
3	2	3
4	3	23
5	4	15
6	5	2
7	6	2
8	7	4
9	8	8
10	9	7
11	10	4
12	11	22
13	12	21

Enter the new Zoning info in this column

If you had an ASCII file with the new data in it, you could use the **Column Options | Import** *option.*

2. In the **ZONE88_NEW.PAT** Table Tool, click the Save icon to save your changes.

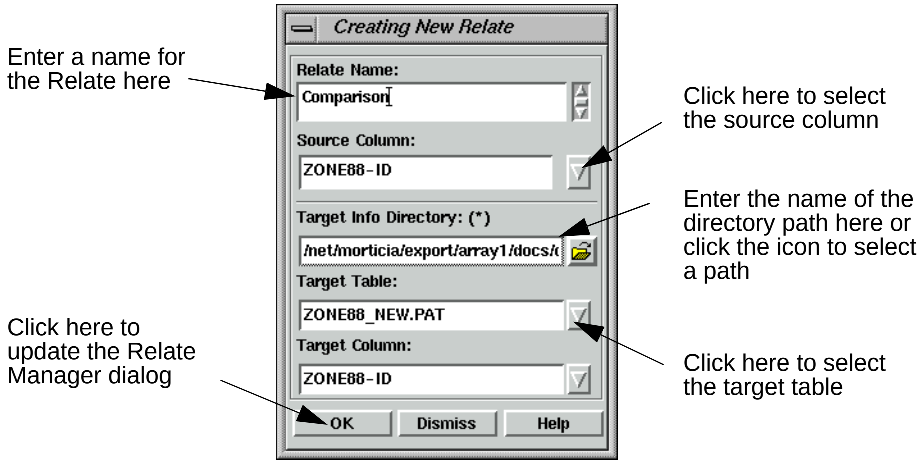
3. In the **ZONE88_NEW.PAT** Table Tool, select **File | Close** from the menu bar.

4. In the **ZONE88.PAT** Table Tool, select **Relate | Relate** from the menu bar.

   The Relate Manager dialog opens.

5. In the Relate Manager dialog, click **New**.

   The Creating New Relate dialog opens.



Relate Name: Comparison

Source Column: ZONE88-ID

Target Info Directory: (*) /net/morticia/export/array1/docs/

Target Table: ZONE88_NEW.PAT

Target Column: ZONE88-ID

Enter a name for the Relate here

Click here to select the source column

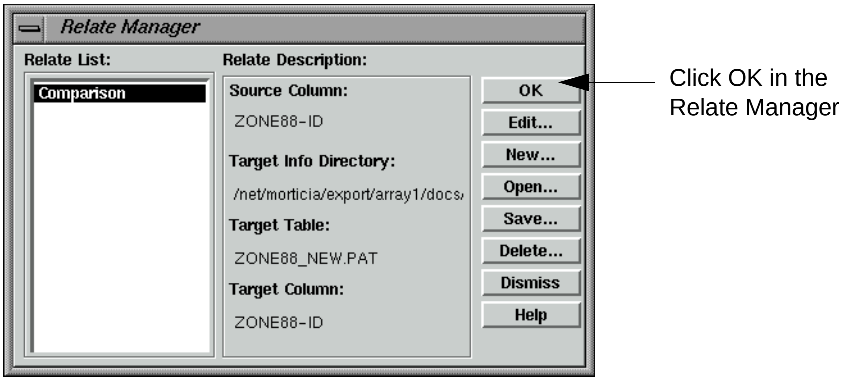
Enter the name of the directory path here or click the icon to select a path

Click here to update the Relate Manager dialog

Click here to select the target table

6. In the Creating New Relate dialog under **Relate Name**, enter the name **Comparison**.
7. Under **Source Column**, click on the popup list and select **ZONE88-ID**.
8. Under **Target Info Directory**, enter the path in which you saved the **ZONE88_NEW.PAT** Table Tool (if it is not already listed).
9. Under **Target Table**, click on the popup list and select **ZONE88_NEW.PAT**.
10. Click **OK** in the Creating New Relate dialog.

    The Creating New Relate dialog closes and the information is displayed in the Relate Manager dialog.



Relate List: Comparison

Relate Description:

Source Column: ZONE88-ID

Target Info Directory: /net/morticia/export/array1/docs/

Target Table: ZONE88_NEW.PAT

Target Column: ZONE88-ID

Click OK in the Relate Manager

NOTE: At this point, you can click **Save** *in the Relate Manager dialog to save the relates to a table for future use.*

11. Click **OK** in the Relate Manager dialog.

    ZONE88.PAT is now related to **ZONE88_NEW.PAT**.

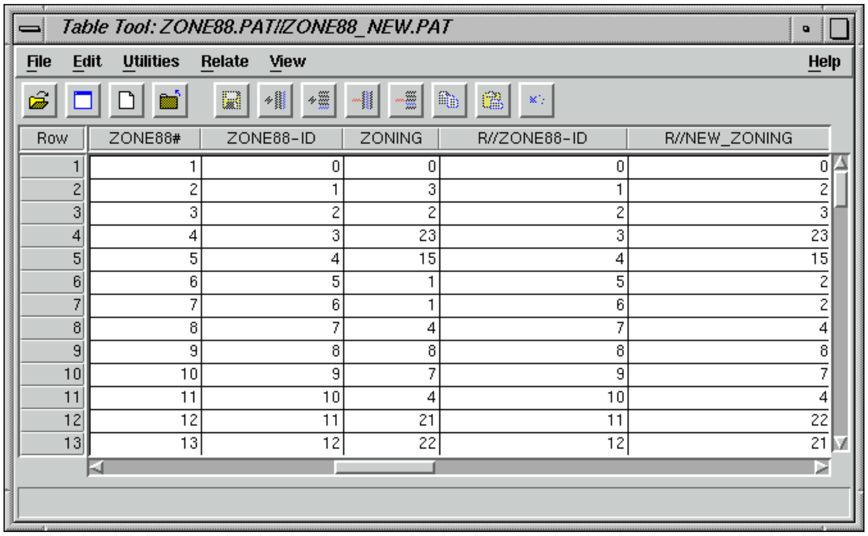


Table Tool: ZONE88.PAT//ZONE88_NEW.PAT
File
Edit
Utilities
Relate
View
Help

Row	ZONE88#	ZONE88-ID	ZONING	R//ZONE88-ID	R//NEW_ZONING
1	1	0	0	0	0
2	2	1	3	1	2
3	3	2	2	2	3
4	4	3	23	3	23
5	5	4	15	4	15
6	6	5	1	5	2
7	7	6	1	6	2
8	8	7	4	7	4
9	9	8	8	8	8
10	10	9	7	9	7
11	11	10	4	10	4
12	12	11	21	11	22
13	13	12	22	12	21

This relationship makes it possible for you to compare the different zoning information.

NOTE: Selecting **Utilities | Table Merge** *enables you to permanently merge the two tables together.*

12. Select **Relate | Drop** in the related Table Tool to drop the related columns.

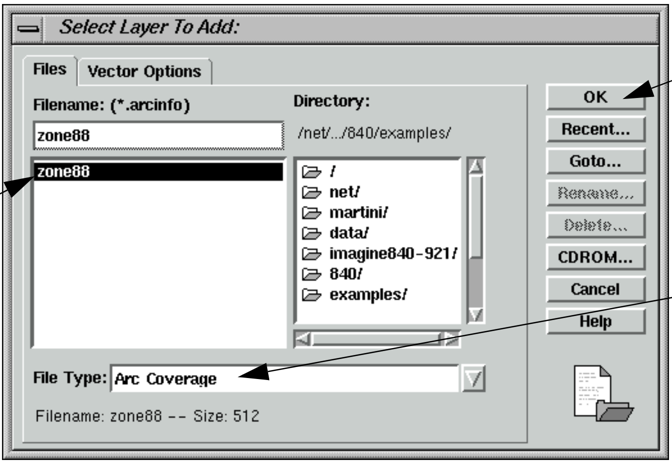
Change Vector Symbology

Display a Layer

ERDAS IMAGINE must be running and a Viewer open.

1. In the Viewer menu bar select **File | Open | Vector Layer**.

The Select Layer To Add dialog opens.


Select Layer To Add:
Files
Vector Options
/
net/
martini/
data/
imagine840-921/
840/
examples/
OK
Recent...
Goto...
Rename...
Delete...
CDROM...
Cancel
Help

Filename: (*.arcinfo) zone88

Directory: /net/.../840/examples/

File Type: Arc Coverage

Filename: zone88 -- Size: 512

Click here to display the vector layer

Click here to select the file

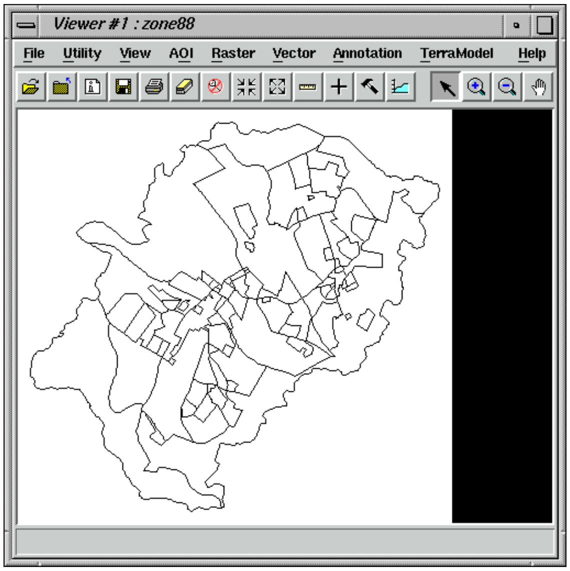
Arc Coverage should be the file type

2. In the Select Layer To Add dialog, confirm that the **File Type** selected is **Arc Coverage**.

3. Under **Filename**, select **zone88**.

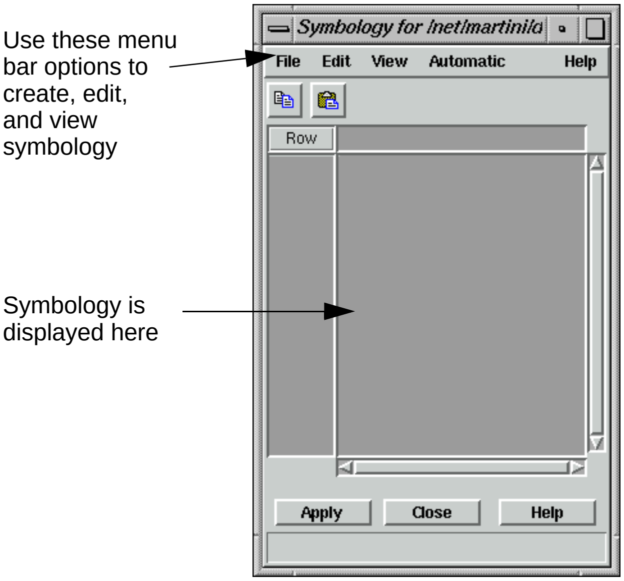
4. Click **OK** in the Select Layer To Add dialog to display the layer in the Viewer.

   The **zone88** polygon layer is displayed in the Viewer, similar to the following example.


Viewer #1 : zone88
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

5. In the Viewer menu bar, select **Vector | Symbology**.

The Symbology dialog opens.


Symbology for /net/martini/a
File
Edit
View
Automatic
Help
Row
Apply
Close
Help

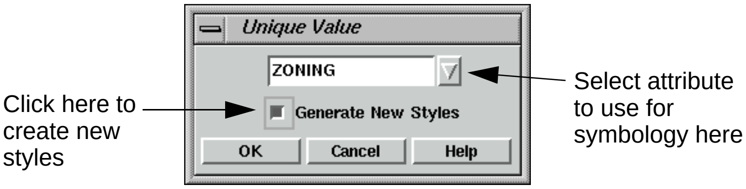
Use these menu bar options to create, edit, and view symbology

Symbology is displayed here

The Symbology dialog has a CellArray, but it does not yet contain any records.

6. In the Symbology dialog, select **Automatic | Unique Value**.

   The Unique Value dialog opens.


Unique Value
ZONING
Generate New Styles
OK
Cancel
Help

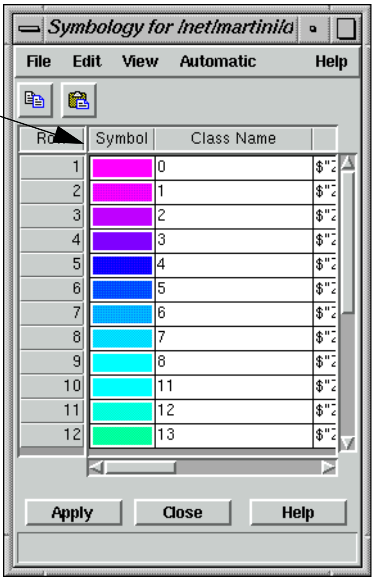
Click here to create new styles

Select attribute to use for symbology here

In this example, you want each zone to have a different symbology.

7. In the Unique Value dialog, click on the popup list and select **ZONING**.
8. Click the **Generate New Styles** checkbox to enable it.
9. Click **OK** in the Unique Value dialog.

   The Unique Value dialog closes and the CellArray of the Symbology dialog fills with the newly generated styles.

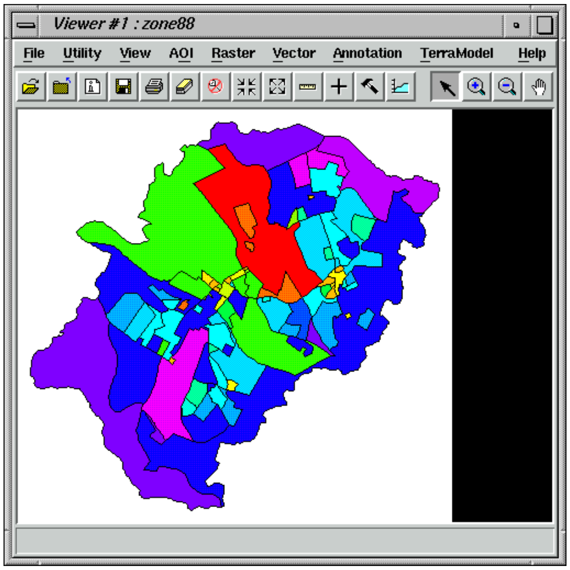

Symbology for /net/martini/d
File
Edit
View
Automatic
Help
Apply
Close
Help

Every Class Name has its own unique symbol

Row	Symbol	Class Name
1	■	0
2	■	1
3	■	2
4	■	3
5	■	4
6	■	5
7	■	6
8	■	7
9	■	8
10	■	11
11	■	12
12	■	13

10. Click **Apply** in the Symbology dialog to apply this new symbology to the displayed layer.

The vector layer is redrawn with the new symbology. As the Symbology CellArray indicates, red areas are zone 23, green are zone 15, dark blue are zone 4, etc.

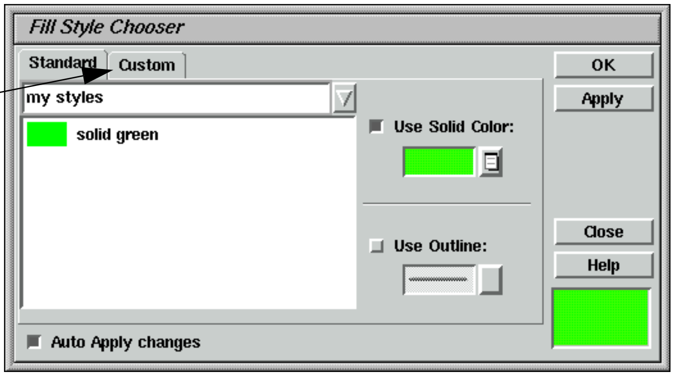

Viewer #1 : zone88
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Add Pattern Polygon Fill

Now, change the style of zone 15 (green) from a plain solid to a pattern fill over a solid.

1. In the Symbology dialog, hold in the **Symbol** column for **Zone 15** (**Row** number **14**) and select **Other**.

   The Fill Style Chooser dialog opens.


Fill Style Chooser
Standard
Custom
OK
Apply
Close
Help
Auto Apply changes

my styles

🟩 solid green

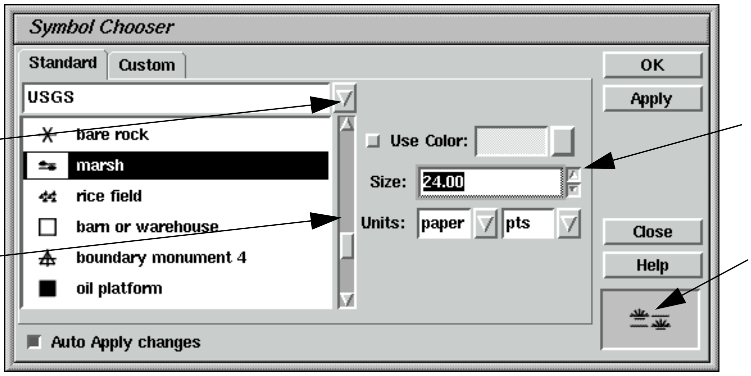
☒ Use Solid Color: 🟩

☐ Use Outline:

Click this tab to select custom style options

2. In the Fill Style Chooser dialog, click the **Custom** tab at the top of the dialog.
3. Click the **Use Pattern** checkbox to activate it.
4. Hold on the **Symbol** popup list and select **Other**.

   The Symbol Chooser dialog opens.


Symbol Chooser
Standard
Custom
OK
Apply
Close
Help
Auto Apply changes

USGS

- bare rock
- marsh
- rice field
- barn or warehouse
- boundary monument 4
- oil platform

☐ Use Color:

Size: 24.00

Units: paper pts

Click on this popup list to select USGS

Use the scroll bar to scroll down the USGS symbol list

Change the symbol size here

A preview of the currently selected symbol displays here

5. Click the popup list and select **USGS** to display the available USGS symbols.
6. Scroll through the symbol list under **USGS** and click on the **marsh** symbol.

   NOTE: This zone is not really marsh; this example is simply to show you how the pattern fill option works.

7. Change the symbol **Size** to **24.00** points.

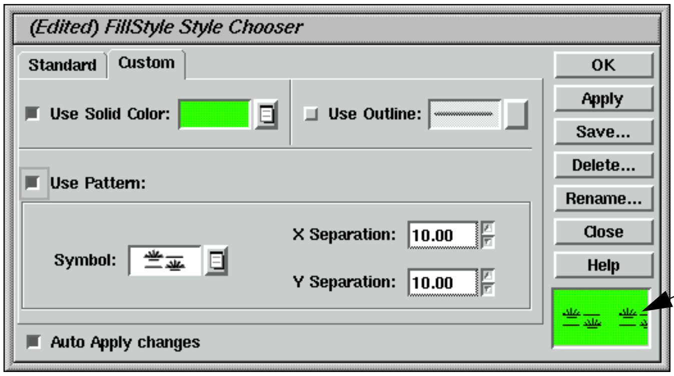
   The preview window in the Symbol Chooser dialog displays the symbol and the symbol size you have selected.

8. Click **Apply** and then **Close** in the Symbol Chooser dialog.

   The preview window in the Fill Style Chooser dialog displays the symbol and the symbol size you selected in the Symbol Chooser dialog.

9. In the Fill Style Chooser dialog, change both the **X Separation** and **Y Separation** to **10.00** points.

   The new style is reflected in the preview window.


(Edited) FillStyle Style Chooser
Standard
Custom
OK
Apply
Save...
Delete...
Rename...
Close
Help

☒ Use Solid Color:

☐ Use Outline:

☒ Use Pattern:

Symbol:

X Separation: 10.00

Y Separation: 10.00

☒ Auto Apply changes

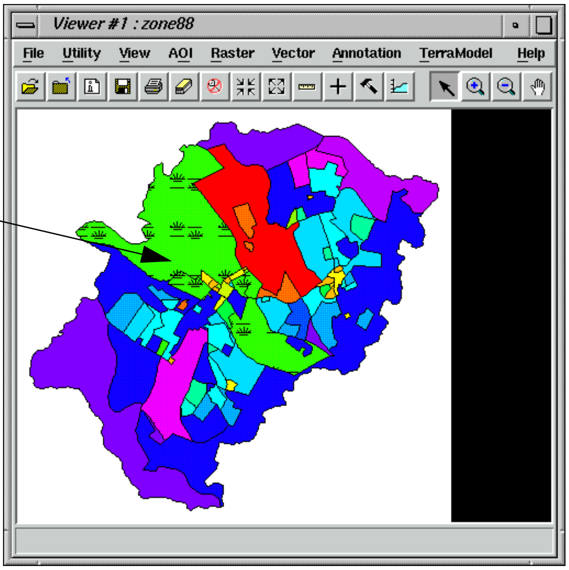
This is the new style

10. Click **Apply** and then **Close** in the Fill Style Chooser dialog.

    The Fill Style Chooser dialog closes and the new style displays in the Symbology dialog.

11. Click **Apply** in the Symbology dialog to apply this new style to the vector layer in the Viewer.

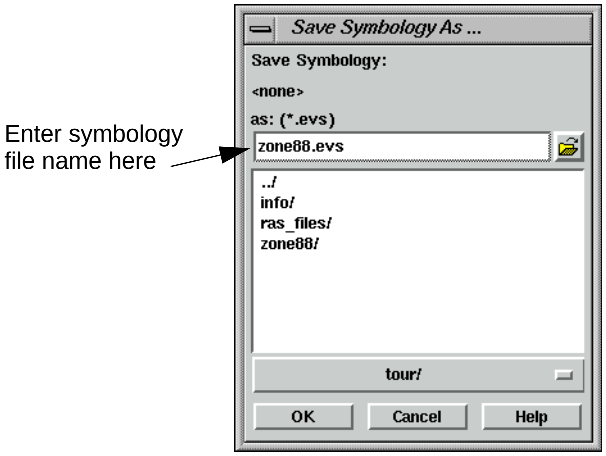
    The polygons in Zone 15 are redrawn with the pattern fill, as in the following example:


Viewer #1 : zone88
File
Utility
View
AOI
Raster
Vector
Annotation
TerraModel
Help

Pattern fill is shown here

12. In the Symbology dialog, select **File | Save As**.

The Save Symbology As dialog opens.



13. In the Save Symbology As dialog, enter a name for the symbology file, such as **zone88.evs**, in the directory of your choice.

If you use the same root name as your coverage, then this is the default symbology file when you open this vector layer later.

14. Click **OK** to save the file.

15. Click **Close** in the Symbology dialog.

16. Select **File | Clear** from the Viewer menu bar.

Build Topology

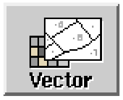
After you have created or edited a vector layer, you must clean or build it to create or maintain the topology and reset the tabular information. If possible, it is always better to run Build instead of Clean. Clean is required only if lines intersect at locations other than nodes, or if there are dangling lines that you wish to automatically delete. This is not the case in this exercise, so you run Build on the new vector layer you just created.

The instructions for running Clean are included at the end of this tour guide for your reference.

Do not clean or build an open vector layer and do not attempt to open a layer that is being cleaned or built.

In order to complete this section, you must have already completed the exercises in "Chapter 4: Vector Querying and Editing" *on page 95. A file you created there,* **zone88subset**, *is used in this example.*

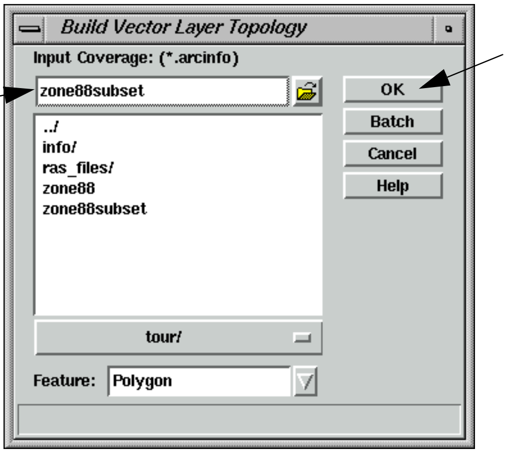
1. Click the Vector icon in the ERDAS IMAGINE icon panel.



The **Vector Utilities** menu opens.

2. Select **Build Vector Layer Topology** from the **Vector Utilities** menu.

   The Build Vector Layer Topology dialog opens.



Input Coverage: (*.arcinfo) zone88subset

Feature: Polygon

Enter vector layer name to build here

Click here to start build

3. In the Build Vector Layer Topology dialog under **Input Coverage**, enter **zone88subset** (or the name you used for the subset in "Chapter 4: Vector Querying and Editing"on page 95), making sure you are in the proper directory.

4. The **Feature** should be **Polygon**.

5. Click **OK** in the Build Vector Layer Topology dialog.

   A Job Status dialog displays, indicating the progress of the function.

6. Click **OK** in the Job Status dialog when the build is complete (unless your Preferences are set to clear the Job Status dialog upon job completion).

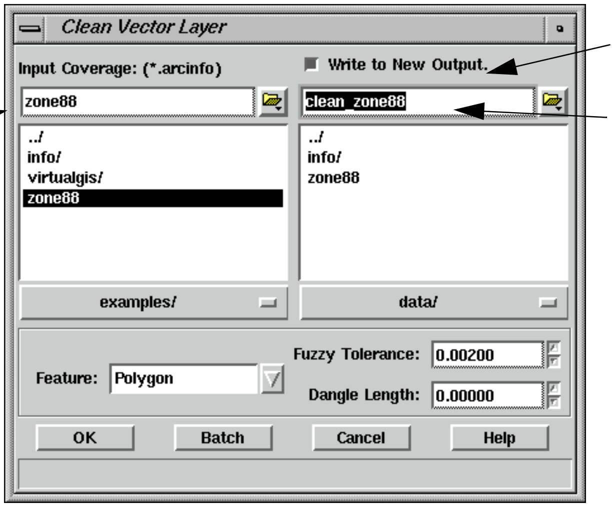
   If you get error messages when trying to display a vector layer in ERDAS IMAGINE, build or clean the layer, then try displaying it again. Do not build or clean an open vector layer and do not attempt to open a layer that is being cleaned or built.

7. Display the vector layer and attribute information to verify that the build ran successfully.

Clean Vector Layer

1. In the **Vector Utilities** menu, click **Clean Vector Layer**.

   The Clean Vector Layer dialog opens.


Clean Vector Layer
../
info/
virtualgis/
zone88
../
info/
zone88
examples/
data/
OK
Batch
Cancel
Help

Input Coverage: (*.arcinfo) zone88

☐ Write to New Output: clean_zone88

Feature: Polygon

Fuzzy Tolerance: 0.00200

Dangle Length: 0.00000

Enter the input coverage name here

Click to write to a new output and then enter the output file name here

2. In the Clean Vector Layer dialog under **Input Coverage**, enter the name of the layer to be cleaned (e.g., **zone88**).

3. Click the **Write to New Output** checkbox and enter a name for the new file. This file name must not be longer than 13 characters.

   NOTE: Although creating a new file is not necessary, it is recommended.

4. The **Feature** should be **Polygon**.

5. For this example, accept the default values for **Fuzzy Tolerance** and **Dangle Length**.

Fuzzy Tolerance and Dangle Length

In general, fuzzy tolerance can be calculated using this formula:

$$\left(\frac{\text{scale}}{\text{\# of inches per coverage unit}}\right) \times 0.002$$

For example, an input source map at a scale of 1:600 with coverage units in feet would have a fuzzy tolerance of .1:

$$\left(\frac{600}{12}\right) \times 0.002 = 0.1$$

The dangle length value is based on the length, in map units, of the longest overshoot that exists in the coverage. The value specified removes any dangling lines that are less than, or equal to, the value you specify. Any lines longer than the length specified remain in the coverage. To measure the dangle length needed for your coverage, you can use the Measurement Tool to measure lines in the coverage.

6. Click **OK** in the Clean Vector Layer dialog.

A Job Status dialog displays, indicating the progress of the function.

7. When the clean is completed, click **OK** in the Job Status dialog (unless your Preferences are set to clear the Job Status dialog upon job completion).

Index

Symbols
.gen file (generic data) 175
.preview.img file (import preview) 163
.sig file (signature file) 456

Numerics
3D Surfacing dialog 282
3X3 Edge Detect 247
3X3 Edge Detect dialog 248

A
Accuracy assessment 496
Accuracy Assessment dialog 496
Add Image icon 61
Add Images for Mosaic dialog 294
Add intermediate hypothesis 566
Add Random Points dialog 498
Affine Model Properties dialog 150
Air photo images 289
AIRSAR 623
Animated Zoom 18
Annotation 69
    borders 69
    color 69
    deselect 82
    edit 92
    grid lines 69, 78
    group elements 78
    legend 69
    line style 69
    lines 80
    scale bar 69
    style 80
        change 80
    symbol 69

- text 69, 80
    - font 69
    - point size 80
- tools
    - grid line 79
    - legend 84
    - map frame 73
    - scale bar 82
    - select map frame 77
    - symbol 89
    - tick mark 69, 79

Annotation Text dialog 86, 89
AOI 22, 456, 472
- selecting signatures 459

AOI Cutline icon 299
AOI Styles dialog 24
AOI tool palette 23, 293, 299, 459
Archive dialog 67
ArcInfo
- vector coverage 95, 627

Area Definition dialog 65
Area Fill dialog 35
Arrange Layers dialog 15, 22, 38, 78, 463, 468, 471
Attention dialog 107
Attribute Options dialog 481
Attributes viewer 101, 110, 112

B

Band Combinations dialog 50
Band-to-color gun assignment 3
Batch
- add multiple files 190
- create new variable 202
- modify commands automatically 203
- modify commands manually 185
- multiple files/multiple commands 193
- multiple files/single command 183, 188
- process later 188
- process now 183
- save/load options 207
- single file/single command 182
- use commands as they are 182

Batch scheduler NT 180
Bicubic Spline 9, 148, 276

Bilinear Interpolation 9, 148, 276
Box Zoom 18
Breakpoint Editor 28
Brightness 27
    adjust 26
Brightness Adjustment dialog 621
Brightness Inversion dialog 253
Build 643
Build Vector Layer Topology dialog 644

C
Camera Model 265
Camera Model Properties dialog 268
Canvas 295
Catalog icon 61
Catalog Image dialog 61
CellArray 37, 101
    add columns 112
    column
        adjust size 40
    Column Options menu 40, 41
    GCP Tool 143
    report 43
    Row Selection menu 40, 101, 103, 345
    select columns 40
    select rows 40
        using criteria 42
    selection criteria 42
    statistics 41
Change Colors 498
Change Colors dialog 498
Chart Options dialog 47
Chip extraction 17
Chip Extraction Viewer 139, 270, 275
Choose AOI dialog 295, 299
Circular Mask dialog 226
Class color
    change 36
Class name
    edit 366
Classification 127, 130, 455, 486, 627
    decision rule 479
        feature space 480
        Mahalanobis distance 480

    - maximum likelihood 480
    - minimum distance 480
    - parallelepiped 480
  - distance file 480
  - output file 480
  - overlay 130, 486
  - supervised 455
  - unsupervised 127, 455, 483
- Classification icon 559
- Classification menu 457, 484, 501, 522, 560
- Classification tools 375
- Classified Image dialog 497
- Classifier icon 457, 484, 491, 501, 522
- Clean 643
- Clean Vector Layer dialog 645
- Coefficient of variation 604, 605
  - calculating 603
- color balancing
  - mosaic 312
- Color Chooser dialog 37, 39, 365
- Color printer 90
- Colorwheel (see Color Selector dialog)
- Column Attributes dialog 112
- Column icon 40
- Column Properties dialog 40, 131, 365, 488
- Composer icon 70, 90, 92
- Compositions dialog 90, 93
- Conditions
  - copy 568
- Constant value 35
- Contingency matrix 472
- Contingency Matrix dialog 472
- Contrast 27
  - adjust 26, 46, 47, 49, 50, 51
  - shift/ bias 31
- Contrast matching 308
- Contrast stretch
  - linear 26
  - piecewise linear 27
- Contrast Tool (piecewise) dialog 27
- Contrast Tool dialog 26
- Convolution dialog 246
- Convolution kernel 352
  - define 354

  - summary 354
- Copy icon 357
- Copy rules and conditions 568
- Copy Vector Layer dialog 96, 629
- Create Feature Space Maps dialog 464
- Create Frame Instructions dialog 73
- Create GCP tool 143
- Create New Signature icon 460
- Create New Table dialog 632
- Create Rule Graphic icon 562
- Create Sensitivity Layer 340
- Creating New Relate dialog 635
- Crisp dialog 249
- Criteria (see Selection Criteria dialog)
- Criteria dialog 377
- Cubic Convolution 9, 148, 276
- Cursor box 17, 74, 76
  - rotate 75
- Cutline 298
- Cutline Selection viewer icon 298

D

- Dangle length 645
- Data
  - ASCII 111
  - display 135
    - preference 3, 59
  - export (see Export)
  - flood plain 347
  - georeferenced 82
  - import (see Import)
  - land cover 347
  - Landsat MSS 3
  - Landsat TM 3, 6, 135
  - raster 627
  - sensitivity 364
    - combine with SPOT 358, 363
  - slope 342
  - SPOT 3
  - SPOT panchromatic 135, 340, 352, 359
    - enhance 352–357
  - subset 343
  - thematic 69
  - transparent 38

- vector 69, 627
  - build 643
  - clean 643
  - copy 96, 111, 628, 629
  - create 109
  - edit 104
    - undo 108
  - select 100, 102
  - style 98
  - symbology 638
- Data Preparation menu 128, 281
- Data View 172, 621, 623
- Data Viewer dialog 172
- DataPrep icon 128, 305
- Default symbol 89
- Defined Areas dialog 65
- DEM 33
  - edit 33
- Dialog xxx
  - 3D Surfacing 282
  - 3X3 Edge Detect 248
  - Accuracy Assessment 496
  - Add Images for Mosaic 294
  - Add Random Points 498
  - Affine Model Properties 150
  - Annotation Text 86, 89
  - AOI Styles 24
  - Archive 67
  - Area Definition 65
  - Area Fill 35
  - Arrange Layers 15, 22, 38, 78, 463, 468, 471
  - Attention 107
  - Attribute Options 481
  - Band Combinations 50
  - Brightness Adjustment 621
  - Brightness Inversion 253
  - Build Vector Layer Topology 644
  - Camera Model Properties 268
  - Catalog Image 61
  - Chart Options 47
  - Choose AOI 295, 299
  - Circular Mask 226
  - Classified Image 497
  - Clean Vector Layer 645

Color Chooser 37, 39, 365
Column Attributes 112
Column Properties 40, 131, 365, 488
Compositions 90, 93
Contingency Matrix 472
Contrast Tool 26
Contrast Tool (piecewise) 27
Convolution 246
Copy Vector Layer 96, 629
Create Feature Space Maps 464
Create Frame Instructions 73
Create New Table 632
Creating New Relate 635
Crisp 249
Criteria 377
Data Viewer 172
Defined Areas 65
Edge Enhancement 608
Export Column Data 111
Export ERDAS 7.5 LAN Data 166
Eye/Target Edit 56
Feature Space to Image Masking 473
Fill Style Chooser 641
Filter Options 231
Form View 64
Formula 132, 489
Fourier Transform 215
Function Definition 349, 355, 360
GCP Matching 146
GCP Tool Reference Setup 275
Generate Script 371
Geo Correction Tools 138, 148, 268
Histogram 476
Histogram Plot Control 476
Hypo Props 561
IHS to RGB 261
Image Files 177
ImageInfo 65, 177, 278, 606
IMAGINE Radar Interpreter module 599
Import Column Data 113
Import Generic Binary Data 172
Import Options 163, 283
Import SPOT 160
Import/Export 160, 165, 621, 623

Indices 259
Inquire Box Coordinates 164, 345
Inquire Color 11
Inquire Cursor 10, 27, 150, 463
Inquire Shape 12
Interpolate 34
Inverse Fourier Transform 222
Job Status 162, 248, 303, 309, 321, 465, 630
Knowledge Base Editor 560
Legend Properties 84
Limits 469
Line Style Chooser 81, 99, 106
Link/Unlink Instructions 20, 367
Linked Cursors 466
List of Recent Files 7
Low/High Pass Filter 219
Map Frame 74, 77
Map Frame Data Source 73
Matching Options 297, 308, 315, 320
Matrix 354
Model Librarian 372
New Column 633
New Map Composition 70
New Vector Layer 109, 114
Open FFT Layer 217
Open Files 492
Open Info Table 631
Options 54, 102, 110
Output Image Options 301
Overlay True Color on Surface 52
Page Setup 373
Polynomial Model Properties 138, 145, 148
Position Parameters 56
Preview Options 161, 162
Print 14, 374
Print Map Composition 91
Projection Chooser 274
Properties 98, 105
Radar Speckle Suppression 602, 604, 612
Raster 340, 342, 345, 347, 350, 354, 377
Read Points 282
Recode 345, 347
Rectangular Mask 228
Red Mouse Linear Mapping 31

Reference GCC File 275
Region Grow Options 461
Region Growing Properties 24, 461
Relate Manager 635
Report Format Definition 43
Resample 148, 151, 277
RGB to IHS 260
Rule Props 563
Run Mosaic 302, 309, 321
Save AOI As 25, 294
Save As 45
Save Layer As 221, 232
Save Model 351, 357, 380
Save Options File 175
Save Signature File As 468
Save Symbology As 643
Scalar 359
Scale Bar Instructions 82
Scale Bar Properties 83
Select A Directory 7
Select Layer To Add 4, 7, 15, 20, 46, 72, 91, 97, 104, 105, 109, 114, 130, 136, 213, 248, 254, 286, 343, 363, 456, 487, 600, 609, 614, 616, 637
Select Profile Tool 46
Selection Criteria 42, 103, 345
Sensor Merge 615
Set Geometric Model 137, 150, 268
Set Grid/Tick Info 79
Set Overlap Function 300, 309, 320
Set Parallelepiped Limits 469
Set Window 342, 380
Shift/Bias Adjustment 30
Signature Alarm 469
Signature Objects 474
Signature Separability 477
Slant Range Adjustment 622
Spectral Statistics 48
Statistics 41, 479
Styles 80
Sun Positioning 54
Supervised Classification 480
Surfacing 285
Symbol Chooser 88, 106, 107, 641
Symbol Properties 89
Symbology 639

  - Tasseled Cap 256
  - Tasseled Cap Coefficients 257
  - Text Properties 87
  - Text String 368, 369
  - Text Style Chooser 81
  - Texture Analysis 618
  - Threshold 491
  - Threshold to File 495
  - Unique Value 639
  - Unsupervised Classification 484
  - Unsupervised Classification (Isodata) 128
  - View Extent 9
  - View Signature Columns 458
  - Viewer Flicker 133, 490, 495
  - Wallis Adaptive Filter 613
  - Wedge Mask 229
- Digitize Points icon 34
- Display options 8
- Display Single icon 64
- Display thematic data 15
- Divergence 477
- Dot Grid Interpretation 542

E

- Edge Enhancement dialog 608
- Electrostatic plotter 90
- Ellipse icon 33
- Ellipse measurement 12
- Enhancement 627
  - in graphical models 352
  - radiometric 252
  - spatial 246
  - spectral 256
- Enterprise Geodatabase 123
- Erase icon 45
- ERDAS Field Guide 279, 599, 622
- ERDAS IMAGINE
  - start 95, 628
- ERDAS IMAGINE icon panel xxi, 70, 92, 128, 159, 340, 457, 484, 491, 501, 522
- ERDAS IMAGINE Viewer 72, 74, 363
- ERS-1 623
- Evaluate 130, 486
- Export

- LAN data 165
- statistics 167

Export Column Data dialog 111
Export ERDAS 7.5 LAN Data dialog 166
Eye/Target Edit dialog 56

F

- Feature space 456
  - to image masking 473
- Feature Space image 464
  - display image 466
- Feature Space to Image Masking dialog 473
- Feature Space viewer 466
- Field of View (FOV) 56
- Fill Style Chooser dialog 641
- Filter 247
  - Prewitt Gradient 609
- Filter Options dialog 231
- Final Analysis Report 555
- Final Analysis Wizard 552
- Form View dialog 64
- Formula dialog 132, 489
- Fourier Analysis menu 215
- Fourier Editor 216
- Fourier Transform
  - create 213
  - Editor 213
    - start 216
    - using tool bar icons 231
- Fourier Transform dialog 215
- Fraction File 557
- Frame area 73
- Frame Sampling Tools 521
- FS to Image Masking 473
- Function Definition dialog 349, 355, 360
- Function icon 300
- Fuyo 623
- Fuzzy tolerance 645

G

- .gen file (generic data) 175
- GCP 17
  - calculating 145
  - delete 144

    - number required 145
    - placement 143
    - select 144
- GCP Matching dialog 146
- GCP Tool 275
    - CellArray 143
- GCP Tool Reference Setup dialog 275
- Generate Script dialog 371
- Geo Correction Tools dialog 138, 148, 268
- Geologic discrimination 618
- Geometric correction 137, 268
- Geometric Correction Tool
    - starting 137
- Georeferencing 135, 265
- Global value 36
- Graphical model 250
    - adjust class colors 364
    - annotation
        - add 368
        - edit 368
        - select 369
        - style 368
    - define function 349, 355, 359
    - define input 342, 354, 359
    - define output 350, 356, 361
    - display output 363
    - edit 251
    - generate text script from 371
    - print 373
    - run 352, 357
    - save 357
    - title 368
    - using scalars in 358
    - view 250
- Graphical Query viewer 62
- Graphics
    - presentation 69
- Grid Generation Tool 534
- Grid Labels 545

H

- High-Pass Filter icon 232
- Histogram 606
- Histogram dialog 476

Histogram graphic 29
Histogram icon 476
Histogram manipulation 28
Histogram Plot Control dialog 476
Hypothesis
    add intermediate 566
    rules for 562
Hypothesis properties dialog 561

I

Icons
    Add Image 61
    AOI Cutline 299
    Catalog 61
    Classification 559
    Classifier 457, 484, 491, 501, 522
    Column 40
    Composer 70, 90, 92
    Copy 357
    Create New Signature 460
    Create Rule Graphic 562
    Cutline Selection viewer 298
    DataPrep 305
    Digitize Points 34
    Display Single 64
    Ellipse 33
    Erase 45
    Function 300
    High-Pass Filter 232
    Histogram 476
    Image Interpreter 215, 239, 246
    Image Matching 297
    Import 159, 629
    Input 296
    Intersection 297
    Line Selection 102
    Lock 34
    Low-Pass Filter 231
    Marquee 102
    Measurement 12
    Modeler 340, 371
    Mouse Linear Mapping 31
    Open 7, 15, 72, 104, 136, 137, 254, 303, 321, 340, 343, 606, 608
    Output 301

    Output Image 301
    Paste 358
    Polyline 13, 50
    Print 374
    Query 63
    Radar 601
    Region Grow 462
    Roam 21
    Run 32, 222, 352, 357
    Save 90, 351, 357
    Select Area for Zooming 62
    Shift/Bias 30
    Statistics 478
    Text 86
    Tools 23, 100, 115
    Vector 95, 628, 643
    Viewer 109, 114, 135, 248, 290, 303, 321, 601
    Visual Query 62
    Window 352
    Zoom In 16
IHS 38
IHS to RGB 260, 261
IHS to RGB dialog 261
Image
    invert 253
    reverse 253
Image Catalog 61
    add information 61
    create 59
Image Commands tool 153
Image Drape viewer 53
Image Enhancement menu 612
Image Files dialog 177
Image Interpreter 9, 237, 339, 495
    functions
        Convolution 608
        Neighborhood 608
Image Interpreter icon 215, 239, 246
Image Interpreter menu 215, 239, 246
Image matching 308
Image Matching icon 297
Image resampling 9
Image Stacking 308
Image Tiles 525

ImageInfo dialog 65, 177, 278, 606
ImageInfo utility 176
    edit 176
IMAGINE Radar Interpreter module
    using the IMAGINE Radar Interpreter module 599
IMAGINE Radar Interpreter module dialogs 599
IMAGINE Radar Interpreter module functions
    Brightness Adjustment 621
    Edge Enhancement 608
    Luminance Modification 611
    Sensor Merge 611
    Slant Range Adjustment 622
    Speckle Suppression 599
    Wallis Adaptive Filter 611
Import
    binary data 171
    column data 113
    decimated image 162
    generic data 171
    Landsat TM 162
    SPOT data 159
    subset 164
Import Column Data dialog 113
Import Generic Binary Data dialog 172
Import icon 159, 629
Import Options dialog 163, 283
Import SPOT dialog 160
Import/Export dialog 160, 165, 621, 623
Index 259
Indices dialog 259
Input icon 296
Input Slope Layer 342
Inquire Box Coordinates dialog 164, 345
Inquire Color dialog 11
Inquire Cursor 150
    style 11
Inquire Cursor dialog 10, 27, 150, 463
Inquire Shape dialog 12
Interpolate dialog 34
Intersection icon 297
Inverse Fourier Transform dialog 222
ISODATA 127, 483

J

Jeffries-Matusita 477
JERS-1 623
Job Status dialog 162, 248, 303, 309, 321, 465, 630

K

Kernel 247
  CellArray 248
  library 247
Keyboard shortcut 7
Knowledge Base
  test 570
Knowledge Base Editor dialog 560
Knowledge engineer 559
  add hypothesis 560
  add intermediate hypothesis 566
  ANDing criteria 583
  copy and edit rules/conditions 568
  enter rules for hypothesis 562
  enter variable for rule 563
  graphic model option 579
  pathway feedback 594
  prompt analyst 592
  set up output classes 559
  test knowledge base 570
  use with spatial logic 585

L

Label point 100
LANDSAT images 304
Legend 84
  columns 85
  position 84, 85
  specify contents 85
Legend Properties dialog 84
Level of Detail 324
Limits dialog 469
Line
  join 108
  reshape 108
  split 108
Line measurement 12
Line of constant range 621
Line Selection icon 102

Line Style Chooser dialog 81, 99, 106
Link box tool 139
Link symbol 20
Link/Unlink Instructions dialog 20, 367
Linked Cursors dialog 466
List of Recent Files dialog 7
Lock icon 34
Lookup table (LUT) graph
    rotate 32
    shift 32
Lookup table (LUT) values
    change 26
    modify 28
    reset 28
Low/High Pass Filter dialog 219
Low-Pass Filter icon 231

M
Magnifier window 16
Magnify image (see Zoom)
Map
    area 73
    create 70
        prepare data layers 72
        steps 69
    edit 92
    file (.map) 92
    name 71
    outside margin 79
    print 90
        to file 91
    save 90
    scale 71
    size 71
    title 86
    units 71, 73
        page 73
Map Composer
    icon 70, 92
    start 70
    viewer 71
Map Composer menu 70, 92
Map frame 73
    adjust size 75

- create 72
- delete 77
- dimensions 73
- edit 77
- position 74, 76
- reference to image in Viewer 76
- scale 74, 76
- select 77
- size 74

Map Frame Data Source dialog 73
Map Frame dialog 74, 77
Map Path Editor 93
Map projection 10, 135, 160, 167, 265
- State Plane 135

Marquee icon 102
Mask 473
Matching Options dialog 297, 308, 315, 320
Matrix dialog 354
Measurement icon 12
Measurement Tool viewer 12
Menu
- Classification 457, 484, 501, 522, 560
- Data Preparation 128
- Fourier Analysis 215
- Image Enhancement 612
- Image Interpreter 215, 239, 246
- Map Composer 70, 92
- Radar 601
- Radar Interpreter 602
- Radiometric Enhance 252
- Session xxi
- Spatial Enhancement 246
- Spatial Modeler 340, 371
- Spectral Enhancement 256
- Tools xxiv
- Utilities xxv
- Vector Utilities 96, 628, 630, 644

Mineral ratio 260
Model
- using conditional statements 349
- working window 342, 380

Model Librarian dialog 372
Model Maker 237, 250, 339
- functions 360

    - Analysis 360
    - Conditional 360
    - CONVOLVE 356
    - Criteria 375
    - EITHER 360
    - STRETCH 360, 361
  - start 340
- Modeler icon 340, 371
- Modify commands automatically 203
- Modify commands manually 185
- Modify variables 198
- Mosaic 289
  - color balancing 312
  - cutline 298
  - display 303
  - input 292
  - intersection 296
  - output 301
  - run 302, 309
- Mosaic Image List 295
- Mosaic Tool
  - viewer 292, 305
- Mouse linear mapping 31
- Mouse Linear Mapping icon 31

N

- Nearest Neighbor 9, 148, 276
- Neatline 78, 79
- New Column dialog 633
- New Map Composition dialog 70
- New variable 202
- New Vector Layer dialog 109, 114
- No Link symbol 21
- North arrow 88

O

- Observer 56
- On-Line Help xxvi
- Opacity 39
- Open FFT Layer dialog 217
- Open Files dialog 492
- Open icon 7, 15, 72, 104, 136, 137, 254, 303, 321, 340, 343, 606, 608
- Open Info Table dialog 631
- Options dialog 54, 102, 110

Orthorectification 265
Output icon 301
Output Image icon 301
Output Image Options dialog 301
Overlay thematic data 52
Overlay True Color on Surface dialog 52

P
.preview.img file (import preview) 163
Page Setup dialog 373
Paragraph styles 522
Paste icon 358
Pattern Polygon Fill 640
Personal Geodatabase 120
Point measurement 12
Polygon Interpretation 541
Polygon measurement 12
Polyline icon 13, 50
Polynomial Model Properties dialog 138, 145, 148
Position Parameters dialog 56
PostScript 374
PostScript device 90
Preference Editor 3, 241
  eml 3
  saving changes 6
  viewer 4
Preview Options dialog 161, 162
Preview window xxx, 33, 38, 86, 88, 641
Prewitt Gradient filter 609
Print dialog 14, 374
Print icon 374
Print Map Composition dialog 91
Prior Data 529
Projection Chooser dialog 274
Properties dialog 98, 105

Q
Query
  coordinates 11
  pixel information 10
Query icon 63
Quick View menu 16, 27, 150, 254

R
Radar icon 601
Radar Interpreter menu 602
Radar menu 601
Radar Speckle Suppression dialog 602, 604, 612
RADARSAT 623
Radiometric Enhance menu 252
Raster Attribute Editor 33, 36, 38, 39, 130, 131, 364, 482, 486, 488, 495
Raster dialog 340, 342, 345, 347, 350, 354, 377
Raster options 290
Rayleigh distribution 603
Read Points dialog 282
Real Time Zoom 19
Recode 340, 345, 347, 496
  using Criteria 345
Recode dialog 345, 347
Rectangle measurement 12
Rectangular Mask dialog 228
Rectification 17
  image to image 135, 265
  steps 135, 265
  verifying 149
Red Mouse Linear Mapping dialog 31
Reference GCC File dialog 275
Region Grow icon 462
Region Grow Options dialog 461
Region Growing Properties dialog 24, 461
Relate Manager dialog 635
Report Format Definition dialog 43
Resample 9, 148, 276
  Bicubic Spline 9, 148, 276
  Bilinear Interpolation 9, 148, 276
  Cubic Convolution 9, 148, 276
  Nearest Neighbor 9, 148, 276
Resample dialog 148, 151, 277
RGB 38
RGB to IHS 260
RGB to IHS dialog 260
RMS error 145
Roam 20
Roam icon 21
Rotate/Magnify Instructions box 105, 114
Rotation box (rot box) 105, 108, 114
Rule Properties dialog 563

Rules
  copy 568
Rules for hypothesis 562
Run icon 32, 222, 352, 357
Run Mosaic dialog 302, 309, 321

S
.sig file (signature file) 456
Sample Node 541
Sample Selection Tool 537
Sampling Grid 529, 534
Sampling Project 522
Save AOI As dialog 25, 294
Save As dialog 45
Save icon 90, 351, 357
Save Layer As dialog 221, 232
Save Model dialog 351, 357, 380
Save Options File dialog 175
Save Signature File As dialog 468
Save Symbology As dialog 643
Scalar dialog 359
Scale 73
Scale bar 82
Scale Bar Instructions dialog 82
Scale Bar Properties dialog 83
Script model
  annotation in 373
  delete 372
  edit 372
  run 372
Select A Directory dialog 7
Select Area for Zooming icon 62
Select Layer To Add dialog 4, 7, 15, 20, 46, 72, 91, 97, 104, 105, 109, 114, 130, 136, 213, 248, 254, 286, 343, 363, 456, 487, 600, 609, 614, 616, 637
Select Profile Tool dialog 46
Selected Samples 529
Selection Criteria dialog 42, 103, 345
Sensor Merge 614
  IHS 615
  Multiplicative 615
  Principal Component 615
Sensor Merge dialog 615
Session Log 159, 604
Session menu xxi, 3

Set Geometric Model dialog 137, 150, 268
Set Grid/Tick Info dialog 79
Set Overlap Function dialog 300, 309, 320
Set Parallelepiped Limits dialog 469
Set up scheduler 180
Set Window dialog 342, 380
Shadow
  enhance 27
Shift/Bias Adjustment dialog 30
Shift/Bias icon 30
Signature
  alarm 469
  contingency matrix 472
  ellipse 475
  evaluate 468
  feature space to image masking 473
  histogram 476
  merge 468
  non-parametric 456, 480
  parametric 456, 480
  separability 477
  statistics 474
Signature Alarm dialog 469
Signature Editor 456, 457
Signature Objects dialog 474
Signature Separability dialog 477
Single Sampling Project Nodes 525
Single Sampling wizard 524
SIR-A 601, 623
SIR-B 623
Slant Range Adjustment dialog 622
Space resection 265
Spatial Enhancement menu 246
Spatial Modeler 237, 495
  Library 371
Spatial Modeler Language 237, 339, 371
Spatial Modeler menu 340, 371
Spatial Profile viewer 49
Speckle noise 599
Spectral Enhancement menu 256
Spectral Profile viewer 46
Spectral Statistics dialog 48
Stacking 308
Stationarity 556

Statistics dialog 41, 479
Statistics icon 478
Stratified Tile 529
Styles dialog 80
Subset 238
Sun Positioning dialog 54
Supervised Classification dialog 480
Surface Profile viewer 51
Surfacing dialog 285
Symbol Chooser dialog 88, 106, 107, 641
Symbol Properties dialog 89
Symbology dialog 639
Symbols 88

T

Tab 214, 456
Table Tool 630
  CellArray 632
Tasseled Cap Coefficients dialog 257
Tasseled Cap dialog 256
Tasseled Cap transformation 256
Test Knowledge Base 570
Text 89
  align 87
  default style 89
  deselect 87
  edit 87
  fonts 86
  position 87
  style 86
Text Editor 44, 473, 478
  ERDAS IMAGINE 372
  menu bar 45
Text entry fields 93
Text icon 86
Text Properties dialog 87
Text String dialog 368, 369
Text Style Chooser dialog 81
Texture 618
Texture Analysis dialog 618
Threshold 481, 491
Threshold dialog 491
Threshold to File dialog 495
Tick label 80

Tick mark 69, 78
  - length 79
  - select 80
  - spacing 79

Tool palette
  - AOI 293, 299, 459
  - Vector 100, 107, 115

Tools icon 23, 100, 115
Tools menu xxiv, 621, 623
Transformation
  - changing the order of 145
  - coefficients 145

Transformation matrix
  - compute 145

Transformed divergence 477

U

Undersampled strata 556
Unique Value dialog 639
Unlink cursor 21
Unsupervised Classification (Isodata) dialog 128
Unsupervised Classification dialog 484
Utilities menu xxv
Utility menu 172

V

Variable 202
Vector icon 95, 628, 643
Vector tool palette 100, 107, 115
Vector Utilities menu 96, 628, 630, 644
Vegetation classification 618
View Extent dialog 9
View Signature Columns dialog 458
Viewer 3
  - Attributes 101, 110, 112
  - Chip Extraction 270, 275
  - Feature Space 466
  - Image Drape 53
  - Measurement Tool 12
  - Mosaic Tool 292, 305

Viewer (ERDAS IMAGINE) 72, 74, 289, 363
  - AOI menu 22
  - arrange layers in 15
  - close 17, 21

  - link 20, 150
  - menu bar 6, 343
  - open new 20
  - Quick View menu 150
  - resize 16, 291
  - split 254
  - title bar 10
  - tool bar 100, 115
  - unlink 21
- Viewer Flicker dialog 133, 490, 495
- Viewer icon 109, 114, 135, 248, 290, 303, 321, 601
- Viewshed Analysis 323
- Viewshed dialog 326, 328
- viewshed layer 327, 329
- Visual Query icon 62

W

- Wallis Adaptive Filter 612
- Wallis Adaptive Filter dialog 613
- Warning box 278
- Wedge Mask dialog 229
- Window icon 352

Z

- Zoom 16
- Zoom In icon 16

GIS & Mapping
Division

geographic imaging made simple℠

Information subject to change without notice.

Copyright © 2002 Leica Geosystems. All rights reserved. ERDAS and ERDAS IMAGINE are registered trademarks; ERDAS Field Guide and ERDAS IMAGINE Tour Guides are trademarks of ERDAS LLC. ERDAS is a wholly-owned subsidiary of Leica Geosystems and operates as the Geographic Imaging unit within Leica Geosystems' GIS & Mapping Division. Other brand and product names are the properties of their respective owners. Imagery courtesy of the city of San Luis Obispo, CA and Golden State Aerial Surveys, DigitalGlobe, i-cubed, and USGS. Part No. EI 8.6 Manuals cc 10/02

Leica Geosystems
GIS & Mapping Division
2801 Buford Highway
Atlanta, Georgia 30329, USA
Phone +1 404 248 9000
Fax +1 404 248 9400
gis.leica-geosystems.com